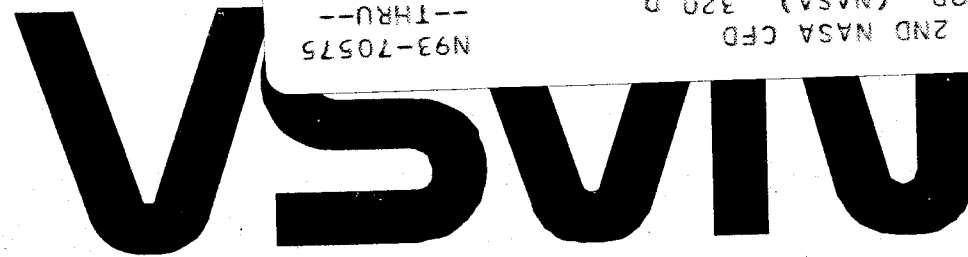


2ND NASA CFD VALIDATION WORKSHOP



NASA Lewis Research Center
July 10-12, 1990

0116670 29/34

N93-70575
--THRU--
N93-70583
Unclass

(NASA-TM-107972) 2ND NASA CFD
VALIDATION WORKSHOP (NASA) 320 p

2320

2ND NASA CFD VALIDATION WORKSHOP

**NASA LEWIS RESEARCH CENTER
CLEVELAND, OHIO 44135**

JULY 10-12, 1990

RAYMOND E. GAUGLER, CHAIRMAN

2ND NASA CFD VALIDATION WORKSHOP
NASA LEWIS RESEARCH CENTER

TABLE OF CONTENTS

	<u>Page</u>
I. Workshop Summary	1
II. Pre-Workshop Activities	
1. Invitation letter	3
2. Conclusions and recommendations of the Advisory Committee, May 1987	5
3. Summary of the 1987 NASA CFD Validation Workshop	9
4. List of attendees	12
III. Workshop Program	15
IV. Workshop Presentations	
Workshop Goals and Agenda	Ray Gaugler 16
LaRC Summary	Scott Kjelgaard 20
ARC Summary	Joe Marvin 45
MSFC Summary	Paul McConnaughey 70
LeRC Summary	Ray Gaugler 100
ARC Technical presentations	William Lockman 126
.	Paul McConnaughey 162
LaRC Technical presentations	Scott Kjelgaard 182
LeRC Technical presentations	Warren Hingst 197
.	Michael Hathaway 232
MSFC Technical presentation	Lisa Griffin 257
V. Small Working Group Reports	
1. Subsonic	292
2. High-Speed	297
3. Hypersonic	303

1990 Workshop Summary

The 2nd NASA CFD Validation Workshop was held at Lewis Research Center on July 10-12, 1990.

The purpose of the workshop was to review NASA's progress in CFD validation since the first workshop (held at Ames in 1987) and to affirm the future direction of the NASA CFD validation program.

Forty-six people participated in the workshop. Of these, 25 were from NASA, 2 were from other government agencies, 13 were from industry, and 6 were from universities. A list of attendees is included in this report.

The first session, held on the morning of the first day, consisted of overviews of CFD validation research at each of the three OAET research centers and at Marshall Space Flight Center. The second session (afternoon of the first day) consisted of in-depth technical presentations of the best examples of CFD validation work at each center (including Marshall).

On the second day the workshop divided into three working groups to discuss CFD validation progress and needs in the subsonic, high-speed, and hypersonic speed ranges. The emphasis of the working groups was on propulsion. The subsonic, high-speed, and hypersonic working groups were led by Gordon Pickett (Pratt and Whitney), Joe Marvin (Ames), and Lou Povinelli (Lewis), respectively. At the end of the second day, each group leader reported to the workshop on his group's findings and recommendations. The rosters of the working groups, and copies of the charts used in reporting out are included in this report.

It was apparent from the presentations that NASA's CFD Validation program has significantly changed the way experimental fluid dynamics research at NASA is performed - even research that is not funded out of the CFD Validation Program. Compared to before the validation program began, researchers today are finding it easier to obtain funding for instrumentation and tunnel time required for code validation. As a result, more experiments are producing data that is useful for code validation. The synergism between experiment and computation is producing better experiments and better computational methods.

CFD Validation Coordinating Board Recommendations

On the third and final day the NASA participants met to discuss the future of the NASA CFD validation program in light of what we had learned in the first two days. The Board, with input from other NASA participants, made the following general recommendations:

1. NASA has made significant progress in implementing the recommendations of the Ad Hoc Committee on CFD validation of the Aeronautics Advisory Committee and should continue to abide by them..

2. NASA has made significant progress in implementing the recommendations of the first NASA CFD Validation Workshop and should continue to follow them.
3. Changes in the work breakdown structure for FY91 and beyond must not adversely affect funding for CFD validation. In particular, since with the new WBS CFD validation will no longer have a funding stream separate from that for CFD Methods, NASA Headquarters and the research centers must ensure that the CFD validation programs is adequately funded.
4. NASA must maintain a highly visible CFD validation program to enhance advocacy for obtaining funding from other sources.
5. NASA should encourage applied programs to incorporate into their test plans methodology that allows for some level of code validation.
6. NASA should advocate to program offices that they provide advanced instrumentation to quantify flow environments.
7. NASA should encourage the continued participation of Marshall Space Flight Center in the validation program. The Board commends their synergistic role in working with the research centers.
8. The Coordinating Board should organize a third NASA CFD Validation workshop to be held at Langley Research Center in approximately three years.

The Board identified the following important propulsion validation needs that are not being adequately addressed:

1. supersonic mixing and combustion ($2 < M < 6$)
 - a. lower speed: instrumentation is lacking
 - b. higher speed: facilities are lacking
2. subsonic combustion
3. transonic afterbody propulsion integration
4. secondary flow systems in propulsion systems
5. supersonic/subsonic jet mixing for HSCT
6. swept-wing aircraft icing

The Board recognized that, since not all cognizant people were in attendance, this list may not be comprehensive.

Sample Invitation Letter

April 23, 1990

Dear:

I would like to invite you or your representative to participate in the Second NASA CFD Validation Workshop, to be hosted by the Lewis Research Center, July 10-11, 1990.

In 1986, NASA's Aeronautics Advisory Committee (AAC) formed an Ad Hoc Committee to review the CFD validation activities at the three NASA Research Centers. The Ad Hoc Committee was charged to: 1) classify CFD research to identify and characterize verification needs and, 2) assess ongoing and planned CFD verification experiments. The ad hoc committee conducted reviews at each of the three NASA research centers and reported their recommendations back to the AAC. In response to those recommendations, NASA conducted the First NASA CFD Validation Workshop, held in the summer of 1987 at the Ames Research Center, as part of the process to insure that the focus and future direction of the validation program was consistent with the anticipated needs of the aerospace community.

The time has now come for a follow-up second workshop. The purpose of this workshop will be to summarize progress in NASA CFD validation experiments, instrumentation, and facilities since the first workshop, and to affirm the future direction of NASA's CFD Validation Program.

It is planned that the first day of the workshop will consist of presentations summarizing the progress in CFD validation at the three research centers and at the Marshall Space Flight Center. The second day will consist of small working group meetings, focusing on specific topics. Since we will be meeting this year at the Lewis Research Center, the emphasis of the meeting will be on propulsion, focusing on the three principle speed ranges - hypersonic, high-speed, and subsonic.

Your active participation in this workshop will be of great value to NASA in insuring that the CFD Validation program continues to make progress in the areas of most critical need.

Please return the enclosed registration form as soon as possible. A detailed program, information on hotels and direction to NASA Lewis will be sent to registrants at a later date.

If you have any questions, please contact the Workshop Chairman, Dr. Raymond E. Gaugler, at (216) 433-5882.

Sincerely,

Neal T. Saunders
Director of Aeronautics

**Conclusions and recommendations of the
Advisory Committee, May 1987**

COMPUTATIONAL FLUID DYNAMICS VALIDATION

Prepared for

NASA'S AERONAUTICS ADVISORY COMMITTEE

by

THE AD HOC COMMITTEE ON CFD VALIDATION
OF THE AERONAUTICS ADVISORY COMMITTEE

May 1, 1987

CONCLUSIONS

Many excellent experimental programs are being conducted at the NASA research centers. The programs are generally attacking pertinent problems. However, only a few of the projects reviewed were considered by the committee to be CFD-validation projects. Further, adequate coordination among NASA centers was not evidenced in the reviews.

CFD validation is a relatively new concept that requires clearly defined and disciplined coupling between CFD and experiments. The committee has proposed the following definition of CFD validation, which emphasizes the computational and experimental discipline required and distinguishes validation from other experimentation related to CFD:

CFD Code Validation. - Detailed surface- and flow-field comparisons with experimental data to verify the code's ability to accurately model the critical physics of the flow. Validation can occur only when the accuracy and limitations of the experimental data are known and thoroughly understood and when the accuracy and limitations of the code's numerical algorithms, grid-density effects, and physical basis are equally known and understood over a range of specified parameters.

The committee recognizes that four categories of experimentation are required for developing CFD capability:

- A. Experiments designed to understand flow physics.
- B. Experiments designed to develop physical models for CFD codes.
- C. Experiments designed to calibrate CFD codes.
- D. Experiments designed to validate CFD codes.

All four categories of tests are important and are necessary to build a mature CFD capability. Validation tests should be only a part of total test focus of NASA and should be formulated to provide specific data for validating CFD codes.

CFD validation is severely hampered in some areas by the lack of critical measurements under realistic conditions- especially at high Mach numbers. Measurements must be taken with adequate accuracy and resolution, and with redundant instruments, in order to evaluate CFD's capability to predict details and trends as well as to explore boundaries of application for specific codes. Specific instrumentation and facilities need to be developed when the state of the art is inadequate.

Equally important for validation is the mapping of the CFD code's sensitivity to numerical algorithms, grid density, and physical models. These effects must be known to the same degree as the experimental accuracy and resolution in order to understand the applicability of the code over a range of flow parameters.

RECOMMENDATIONS

It is recommended that

1. NASA classify its CFD related experiments into the above-defined categories A, B, C, and D. The classification provides a framework for future evaluation and allocation of resources.
2. NASA adopt a sharply defined program plan for CFD validation, consistent with the definitions proposed by the Committee in this report. In particular,
 - o Projects falling under this program should be closely coordinated and adequately funded.
 - o The limitations of the facilities, instrumentation, proposed models, and codes should be clearly specified at the onset of the projects, as should the critical data set required and the range of validity of the modeling assumptions.
 - o The questions of such phenomenon as unsteadiness, three-dimensionality, transition, turbulence models, and the applicability of inviscid/interacting-viscous/Navier-Stokes models should influence the choices of experiments, code development, and application.
 - o Duplication of experiments and code development should involve several centers.
 - o A validated code should become a NASA or general code, not merely the code of a given center.
3. NASA pay careful attention to the instrumentation and facilities required to provide data of adequate accuracy and in the form and detail required for CFD validation. Close coordination must be maintained between centers.
4. NASA establish a Coordinating Board for CFD validation, consisting of key project leaders from each of the three NASA centers and chaired by NASA Headquarters. This board should have responsibility for defining, coordinating, and focusing the CFD validation effort.

Summary of the 1987 NASA CFD Validation Workshop

1987

WORKSHOP SUMMARY

The first NASA CFD Validation workshop was held at the Ames Research Center on July 14-16, 1987 with 108 persons in attendance. After being welcomed by the Director of the Office of Aerophysics at the Ames Research Center, NASA Headquarters management presented introductory comments and expectations for the meeting. Recommendations from a recent review of CFD validation activities at three NASA Research Centers, conducted by the Ad Hoc Committee on CFD Validation of the Aeronautics Advisory Committee (AAC), were presented. Definitions developed by the Committee for code verification, code calibration and corresponding experiments were also introduced and discussed. NASA will use these definitions in conjunction with the CFD Validation program as a tool to manage and maintain a balanced program. For completeness, a copy of the AAC Ad Hoc Committee report is included herein.

The Ames, Langley, and Lewis Research Centers presented overviews of their CFD Validation activities with brief descriptions of their key experiments. These presentations covered the entire speed range and included external and internal aerodynamic flows, combustion, aerothermal loads, and aeroelasticity. Following the NASA presentations, other government installations, industry, and universities presented overviews of their CFD Validation activities and needs. The presentations and speakers are listed in the Workshop Program which is included herein along with copies of the presentations.

After these formal presentations, the workshop attendees were divided into six working groups. The working groups focused on the following key areas: Low Speed Aerodynamic Flows; High Speed Aerodynamic Flows-Commercial; High Speed Aerodynamic Flows-Military; Internal Flows; Hypersonics-Chemically Reacting External/Internal Flows; and Hypersonics-External Aerothermodynamics. Each working group was given a list of five issues to address in their discussions. The issues were: (1) identify key efforts (numerical and experimental) required to meet immediate modeling and validation needs; (2) identify near-term and far-term critical problem areas that require new or additional modeling and validation activities; (3) identify computational, experimental, facility and instrumentation, or other capabilities required to investigate these critical problem areas; (4) identify key modeling and validation projects that are potential cooperative/joint ventures; and (5) define, identify and/or propose standardized test cases for modeling and validation.

The working groups met for about 6 hours to discuss these issues and prepare summaries.

The formal workshop resumed with the various spokespersons presenting the results from their group discussions. These presentations have also been included herein. Several common recommendations were made. These, not in priority order, included: (1) provide closer cooperation between CFD developers and experimentalists at the outset of verification projects with a lasting commitment from both to see the projects through to completion; (2) provide detailed measurements of the flow field and boundary conditions in addition to model surface measurements and integral quantities; (3) provide improved or new non-intrusive measurement capabilities, especially for

hypersonic or reacting flow conditions; (4) provide redundancy in both measurements and experiments whenever practical so as to clarify data accuracy and credibility; (5) provide dedicated large facilities for validation research and increase flight-based research activities; and (6) provide standardized test cases with accessible electronic data bases. The recommendation for standardized test cases received a strong consensus during the workshop. NASA will review the recommendations from the groups and determine a course of action to advocate their implementation.

The workshop concluded with the general feeling that the meeting was very successful. Attendees from the various research centers, universities and industry expressed a renewed understanding of each other's viewpoints on CFD validation. Experts in computational and experimental fluid dynamics were able to begin the synergistic interaction critical to successful validation activities in their one-on-one discussions. Many felt that this workshop should be the forerunner of future workshops on specific topic areas because the workshop atmosphere provides a forum for discussing both successes and failures which taken together often lead to more expeditious problem solutions.

2nd NASA CFD Validation Workshop
NASA Lewis Research Center
July 10-12, 1990

ATTENDEES

Dr. John Adamczyk
National Aeronautics & Space Admin-
istration
Lewis Research Center
21000 Brookpark Rd., M. S. 5-9
Cleveland, OH 44135

Mr. Kyung Ahn
National Aeronautics & Space Admin-
istration
Lewis Research Center
21000 Brookpark Rd., M. S. 301-2
Cleveland, OH 44135

Mr. Essam Atta
General Electric Aircraft Engines
1 Neumann Way MD/A323
Cincinnati, Ohio 45215

Dr. I. L. Bhateley
General Dynamics, Fort Worth Divi-
sion
P. O. Box 748, Mail Zone 2870
Fort Worth, TX 76101

Dr. William B. Compton
National Aeronautics & Space Admin-
istration
Langley Research Center
M.S. 280
Hampton, VA 23665-5225

Dr. Michael E. Crawford
University of Texas
Mechanical Engineering Dept.
Austin, TX 78712

Dr. Don Dietrich
General Electric Aircraft Engines
1 Neumann Way M.D. A-317
Cincinnati, OH 45215

Dr. J. Philip Drummond
National Aeronautics & Space Admin-
istration
Langley Research Center
M.S. 156
Hampton, VA 23665-5225

Mr. Kevin Early
General Electric Aircraft Engines
1 Neumann Way M.D. A-317
Cincinnati, OH 45215

Dr. Alan H. Epstein
Massachusetts Institute of Technolo-
gy
Gas Turbine Laboratory
Dept. of A. & A., 31-266
Cambridge, MA 02139

Dr. Raymond E. Gaugler
National Aeronautics & Space Admin-
istration
Lewis Research Center
M.S. 5-11
Cleveland, OH 44135

Dr. Steve Gegg
Allison Gas Turbine
Division of GMC
P.O. Box 420 (S-46)
Indianapolis, IN 46206

Professor F. B. Gessner
Department of Mechanical Engineering
University of Washington
M/C FU-10
Seattle, WA 98105

Ms. Lisa W. Griffin
National Aeronautics & Space Admin-
istration
George C. Marshall Space Flight
Center
M.S. ED32
Marshall Space Flight Center, AL
35812

Dr. Michael Hathaway
National Aeronautics & Space Admin-
istration
Lewis Research Center
21000 Brookpark Rd., M. S. 5-11
Cleveland, OH 44135

Dr. Warren Hingst
National Aeronautics & Space Admin-
istration
Lewis Research Center
21000 Brookpark Rd., M. S. 5-11
Cleveland, OH 44135

Mr. Michael Holden
CALSPAN Corp.
4455 Genesee Street
Buffalo, NY 14221

Dr. Mounir B. Ibrahim
Cleveland State University
Mechanical Engineering Dept.
Cleveland, OH 44115

Mr. Anthony M. Ingraldi
National Aeronautics & Space Admin-
istration
Langley Research Center
M.S. 280
Hampton, VA 23665-5225

Mr. Scott O. Kjelgaard
National Aeronautics & Space Admin-
istration
Langley Research Center
M.S. 347
Hampton, VA 23665-5225

Mr. William K. Lockman
National Aeronautics & Space Admin-
istration
Ames Research Center
M.S. 230-2
Moffett Field, CA 94035-1000

Mr. John B. Malone
National Aeronautics & Space Admin-
istration
Langley Research Center
M.S. 173
Hampton, VA 23665-5225

Mr. Joseph G. Marvin
National Aeronautics & Space Admin-
istration
Ames Research Center
M.S. 229-1
Moffett Field, CA 94035

Dr. Paul McConnaughey
National Aeronautics & Space Admin-
istration
George C. Marshall Space Flight
Center
M.S. ED32
Marshall Space Flight Center, AL
35812

Dr. Edward Mularz
National Aeronautics & Space Admin-
istration
Lewis Research Center
21000 Brookpark Rd., M. S. 5-11
Cleveland, OH 44135

Dr. Gordon F. Pickett
Pratt & Whitney
Aircraft Group
400 Main St. M.S. 163-01
East Hartford, CT 06108

Dr. Mark G. Potapczuk
National Aeronautics & Space Admin-
istration
Lewis Research Center
M.S. 77-10
Cleveland, OH 44135

Dr. Louis A. Povinelli
National Aeronautics & Space Admin-
istration
Lewis Research Center
21000 Brookpark Rd., M. S. 5-11
Cleveland, OH 44135

Dr. Lonnie Reid
National Aeronautics & Space Admin-
istration
Lewis Research Center
21000 Brookpark Rd., M. S. 5-11
Cleveland, OH 44135

Professor Eli Reshotko
Case Western Reserve University
Cleveland, OH 44106

Dr. Theodore A. Reyhner
Boeing Commercial Airplane Group
P. O. Box 3707, M.S. 7W-68
Seattle, WA 98124-2207

Ms. Pamela Richardson
National Aeronautics & Space Admin-
istration
Code RF
Washington, DC 20546

Dr. Edward Schairer
National Aeronautics & Space Admin-
istration
Code RF
Washington, DC 20546

Mr. Hyoun-Woo Shin
General Electric Aircraft Engines
1 Neumann Way MD/A323
Cincinnati, Ohio 45215

Mr. Frederick F. Simon
National Aeronautics & Space Admin-
istration
Lewis Research Center
21000 Brookpark Rd., M. S. 5-11
Cleveland, OH 44135

Dr. Terrance W. Simon
University of Minnesota
Dept. of M.E. - 125 M.E. Bldg.
111 Church Street, S.E.
Minneapolis, MN 55455

Dr. Robert Simoneau
National Aeronautics & Space Admin-
istration
Lewis Research Center
21000 Brookpark Rd., M. S. 5-11
Cleveland, OH 44135

Dr. Frank W. Spaid
McDonnell Douglas Research Laborato-
ries
P.O. Box 516, 222/1111101
St. Louis, MO 63166-0516

Dr. Anthony J. Strazisar
National Aeronautics & Space Admin-
istration
Lewis Research Center
21000 Brookpark Rd., M. S. 5-11
Cleveland, OH 44135

Mr. Walter B. Sturek
USA Ballistic Research Laboratory
ATTN: SLCBR-LF-C
Aberdeen Proving Ground, MD 21005-
5066

Dr. S. V. "Mani" Subramanian
General Electric Aircraft Engines
1 Neumann Way M.D. A-319
Cincinnati, OH 45215

Mr. Kenneth Suder
National Aeronautics & Space Admin-
istration
Lewis Research Center
21000 Brookpark Rd., M. S. 5-11
Cleveland, OH 44135

Dr. Wei W. Tseng
Naval Air Development Center
Code 6051
Warminster, PA 18974

Dr. Irwin E. Vas
Boeing Defense and Space Group
Aerospace and Electronics Division
499 Boeing Blvd, M.S. JX-23
P.O. Box 240002
Huntsville, AL 35824-6402

Dr. August Verhoff
McDonnell Aircraft Co.
MC 0341260
P.O. Box 516
St. Louis, MO 63166

Mr. Jerry R. Wood
National Aeronautics & Space Admin-
istration
Lewis Research Center
21000 Brookpark Rd., M. S. 5-11
Cleveland, OH 44135

2nd NASA CFD VALIDATION WORKSHOP
NASA LEWIS RESEARCH CENTER
ADMINISTRATION BUILDING AUDITORIUM
JULY 10-12, 1990

PROGRAM

Tues., July 10, 1990

8:30	Registration, Administration Building Auditorium	
9:00	Welcome	Lonnie Reid
9:10	Workshop Goals and Agenda	Ray Gaugler
9:20	Overview of NASA CFD Validation Program	Ed Schairer
9:45	LaRC Summary	Scott Kjelgaard
10:15	ARC Summary	Joe Marvin
10:45	Break	
11:00	MSFC Summary	Paul McConnaughey
11:45	LeRC Summary	Ray Gaugler
12:15	Lunch - On your own, Lewis Cafeteria	
1:15	ARC Technical presentations	William Lockman
		Paul McConnaughey
2:15	LaRC Technical presentations	Scott Kjelgaard
3:15	Break	
3:30	LeRC Technical presentations	Warren Hingst
		Michael Hathaway
4:30	MSFC Technical presentation	Lisa Griffin
5:00	Social Hour, Hors D'Oeuvres & Cash Bar, Ad. Building Foyer	

Wed., July 11, 1990

8:00	Instructions to working groups	Ray Gaugler
8:15	Meeting of working groups	
12:15	Lunch - On your own, Lewis Cafeteria	
1:15	Meeting of working groups	
3:30	Working groups report	
4:30	Adjourn	

Thurs., July 12, 1990

8:00	Government caucus
12:00	Adjourn

2ND NASA CFD VALIDATION WORKSHOP
JULY 10-12, 1990



2nd NASA CFD VALIDATION WORKSHOP

**National Aeronautics and
Space Administration**
Lewis Research Center

GOALS

- **SUMMARIZE PROGRESS SINCE '87 WORKSHOP**
- **AFFIRM FUTURE DIRECTION OF NASA'S CFD VALIDATION PROGRAM**

51-34
116671 P25

CFD Valid Workshop

Langley Research Center CFD Code Validation Program Overview

Scott O. Kjelgaard

Experimental Methods Branch

Fluid Mechanics Division

Second NASA CFD Validation Workshop

NASA - Lewis Research Center

July 10-12, 1990

N93-70576

Outline

- LaRC Approach to CFD Code Validation
- Experimental/CFD Perceptions
- CFD Code Validation Program Experiment Overview
- Experiment Highlights
- Concluding Remarks

CFD Code Validation

Objective: Conduct detailed benchmark experiments yielding high-quality archival data bases. Use these data bases to validate CFD codes and develop empirical models of complex fluid dynamic phenomena

Approach:

Conduct cooperatively-designed experiments which provide:

- detailed flowfield and surface flow measurements required by code developers
- assessment of measurement errors
- redundant measurements with more than one instrument, in more than one facility
- documented results with archival storage of data for easy accessibility

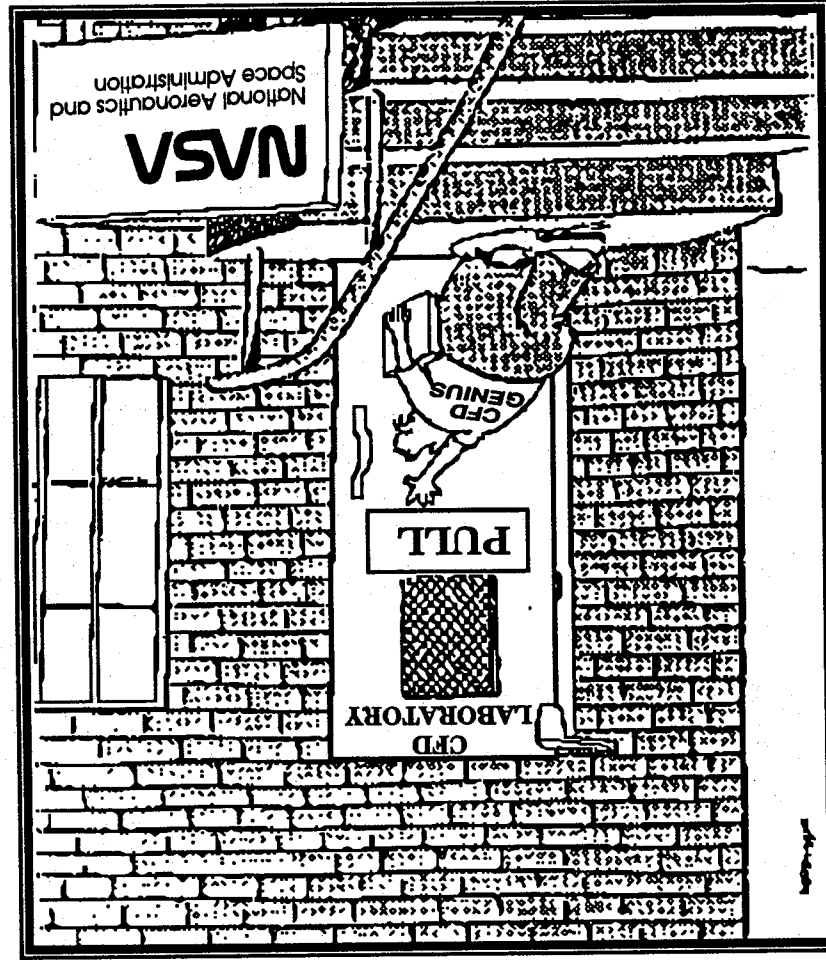
Acquire dedicated, advanced instrumentation systems for the primary code validation facilities

Develop new instrumentation techniques when required

THE CFDERS VIEW OF EXPERIMENTALISTS?



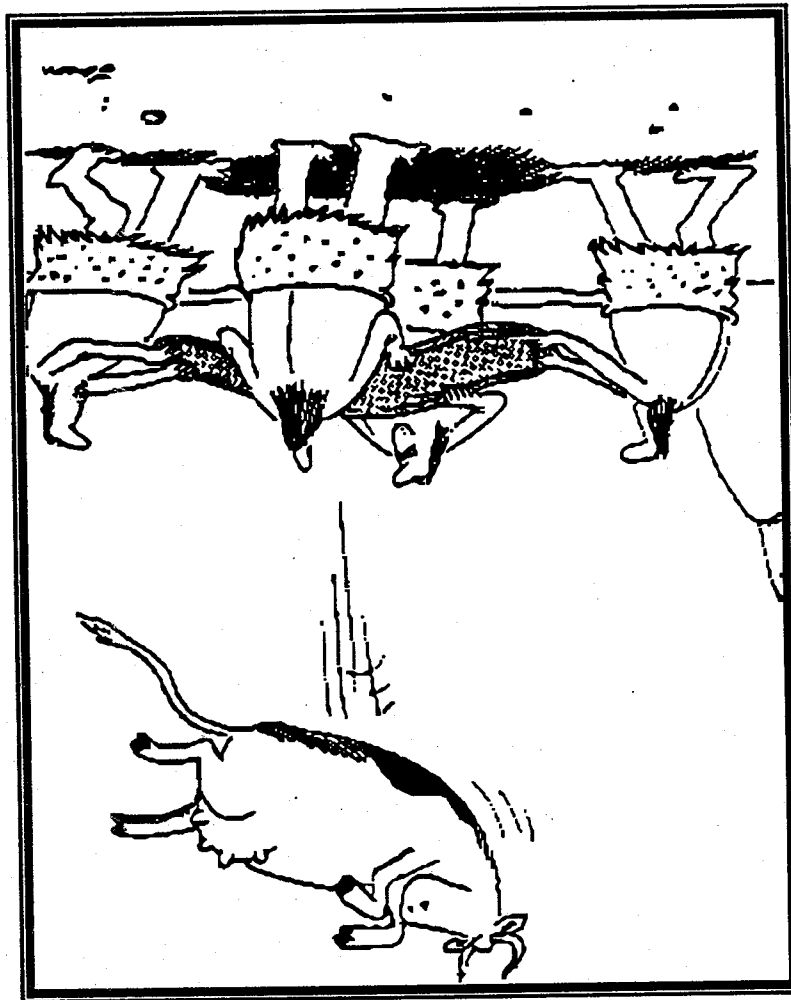
THE EXPERIMENTALIST'S VIEW?

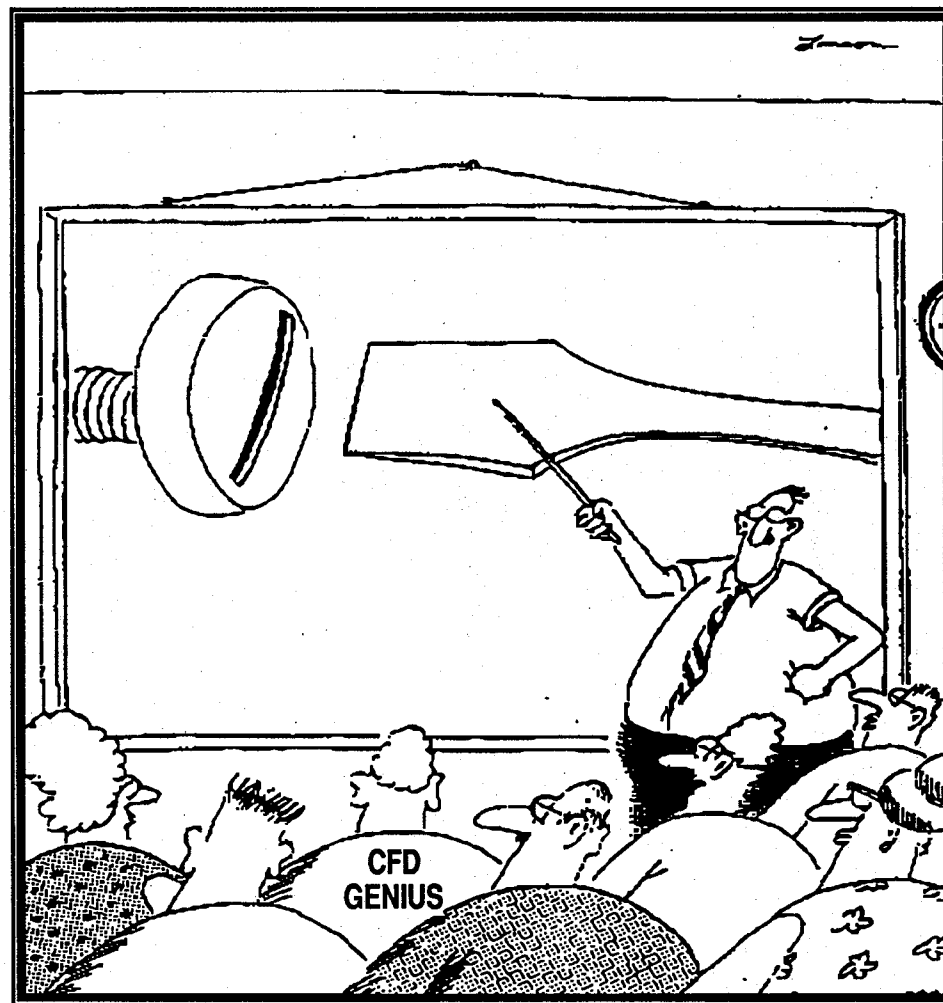


25

July 1990

EXPERIMENTALISTS ATTEMPT FIRST MILKSHAKE





**EXPERIMENTAL TECHNIQUES CLASS
FOR CFD'ERS**

CFD Code Validation Experiments

LaRC Code Validation Program

- 65° Swept Delta Wing
- 76° Swept Delta Wing
- Vertical Tail/Vortex Interaction Investigation
- Rearward Facing Step Experiment
- Particle Imaging Velocimeter (PIV)
- Doppler Global Velocimeter (DGV)
- Holocinematographic Velocimeter (HCV)
- LTPT Laser Velocimeter

Other Programs

- F-18 Mean Flowfield Measurements
- Transonic After-Body Experiment
- Supersonic Coaxial Jet

76° Delta Wing Experiment

Objective:

Develop a detailed data base for the flow over sharpened edged delta wings with a leading-edge sweep of 76°. Use three component LV to obtain flowfield measurements in the burst vortex over the delta wing.

Approach:

Conduct experiments in various facilities documenting the force and moments, surface pressures and flowfield velocities throughout the angle-of-attack range.

Data acquired:

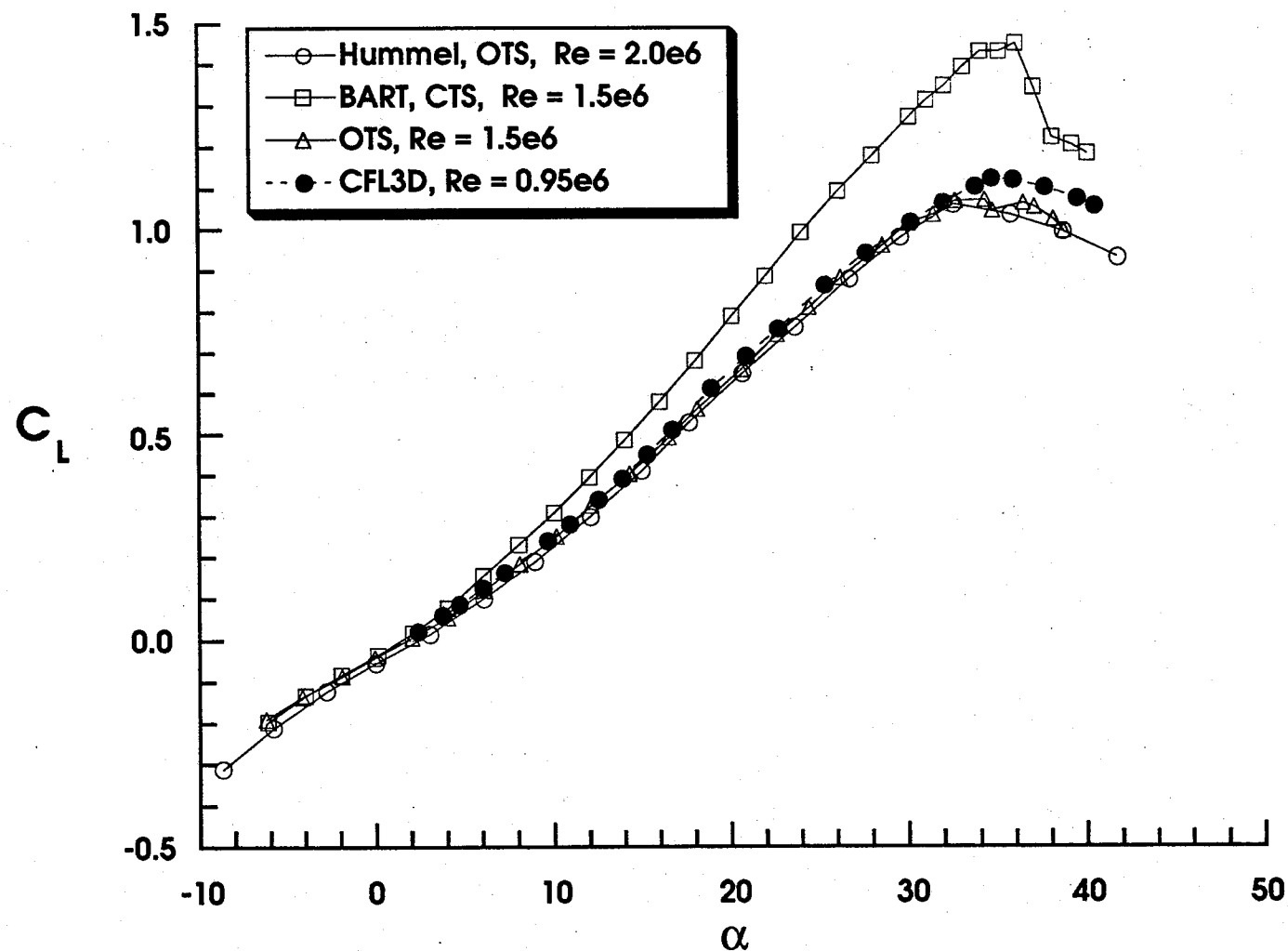
- Forces, Moments
- Static and Dynamic Surface Pressures
- Three component LV, 5-Hole Probe

Facility

BART, 7x10, Vigyan
BART
BART

76° Delta Wing Experiments

Wind Tunnel Comparison



Vertical Tail/Vortex Interaction Investigation

Objective:

- **Provide a fundamental understanding of the vortex-fin interaction process**
- **Determine the capability of CFD methods to predict the physics of the flowfield**
- **Provide a data base for developing improved tail buffet criteria**
- **Develop experimental measurement techniques**

Vertical Tail/Vortex Interaction Investigation

Approach:

- Cooperative MCAIR-Langley MOA in place to study vortex flows and vortex-fin interactions
- Multi-disciplinary program involving structural dynamics and aerodynamics groups at MCAIR and Langley
- Experiments to be conducted using a simple delta wing models with twin vertical tails

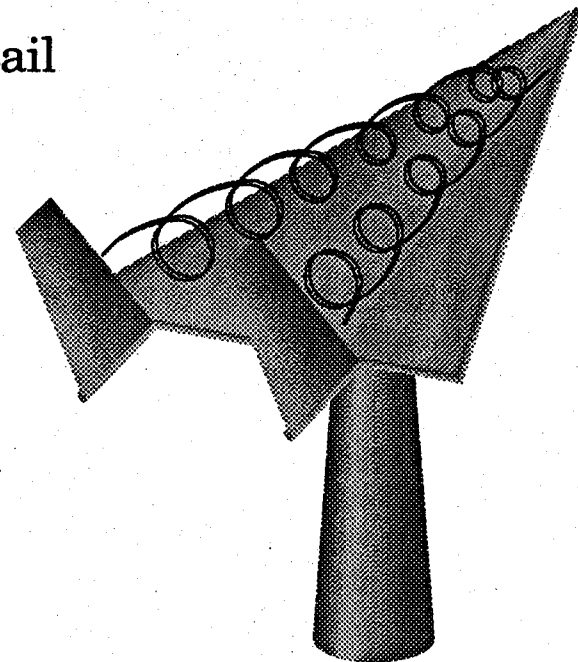
Vertical Tail/Vortex Interaction

Experimental Set-up:

- 76° delta wing using Hummel planform
- 3 component LV measurements to determine the aerodynamic input forcing function to the vertical tails
- Unsteady surface pressure measurements on the rigid tail to be correlated with the velocity fluctuations in the vortex
- Dynamic response from the flexible tail

Measurement Techniques

- Unsteady Surface Pressures
- Tail Bending and Torsion
- Laser Light Sheet Flow Visualization
- 3 component Laser Velocimeter



65° Delta Wing Experiment

Objective:

Develop a detailed data base for the flow over rounded-edge delta wings with a leading-edge sweep of 65°. Measurements will be obtained throughout the Mach and Reynolds number range in LTPT, NTF, and 8' TPT at LaRC.

Low Turbulence Pressure Tunnel

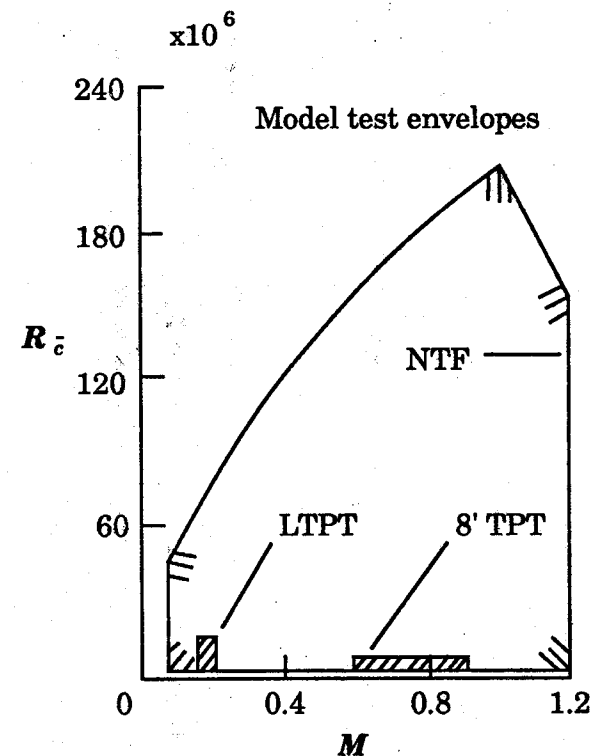
- Static surface pressures
- Flow visualization
- 3 component LV

8-foot Transonic Pressure Tunnel

- Static surface pressures
- Flow visualization
- 3 component LV

National Transonic Facility

- Static surface pressures
- Forces, moments, buffet
- Vortex-core condensation



Geometric Features of NTF Delta Wing

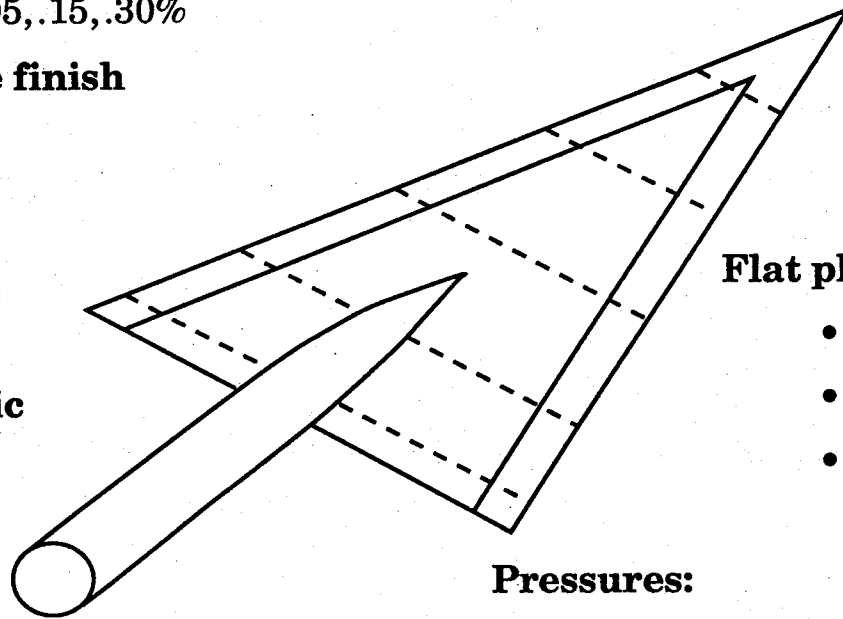
$$\Lambda_{le} = 65^\circ, c_r \approx 2.14 \text{ ft}, b = 2.0 \text{ ft}$$

Interchangeable leading Edges

- $[r_{le}/\bar{c}] s = .05, .15, .30\%$
- 8μ surface finish

Sting

- Symmetric
- $d/b = 0.14$



Flat plate center wing

- $t/c_r = 0.034$
- 8μ surface finish
- Sharp trailing edge

Pressures:

- 183 orifices / configuration
- $x/c_r = .2, .4, .6, .8, .95$
- every 10% x/c_r along l.e.
- 0.010 inch diameter

Particle Imaging Velocimeter (PIV)

Objective:

Develop non-intrusive measurement techniques to obtain the data required for CFD code validation

Approach:

Particle trajectories are tracked by double pulsing a laser light sheet and recording the images. Particles in the sheet form a double-spot pattern on the image when illuminated by the laser beam. The spacing and orientation of the spot-pairs provide an instantaneous measurement of the velocity in a two-dimensional plane in the flowfield

Photograph of the Particle Image Velocimetry System

CFD Valid Workshop

July 1990

7/9/90

17

36

**Photograph of the
Particle Image Velocimetry
Data Analysis**

75° Delta Wing Investigation, $\alpha = 20.5^\circ$

Flow Visualization

- Surface using TiO₂
- Off-body using scanning laser light sheet

Flowfield Surveys

- *Pitot pressure*

$x/L = 0.3, 0.5, 0.7, 0.9, 1.1$, $Re = 0.5, 1.0, 1.5$ million

Each survey plane contains ≈ 2800 data locations

- *Velocity (5-Hole Probe)*

$x/L = 0.7, 0.9, 1.1$; $Re = 0.5, 1.0, 1.5$ million

Each survey plane contains ≈ 3400 data locations

- *Velocity (LV)*

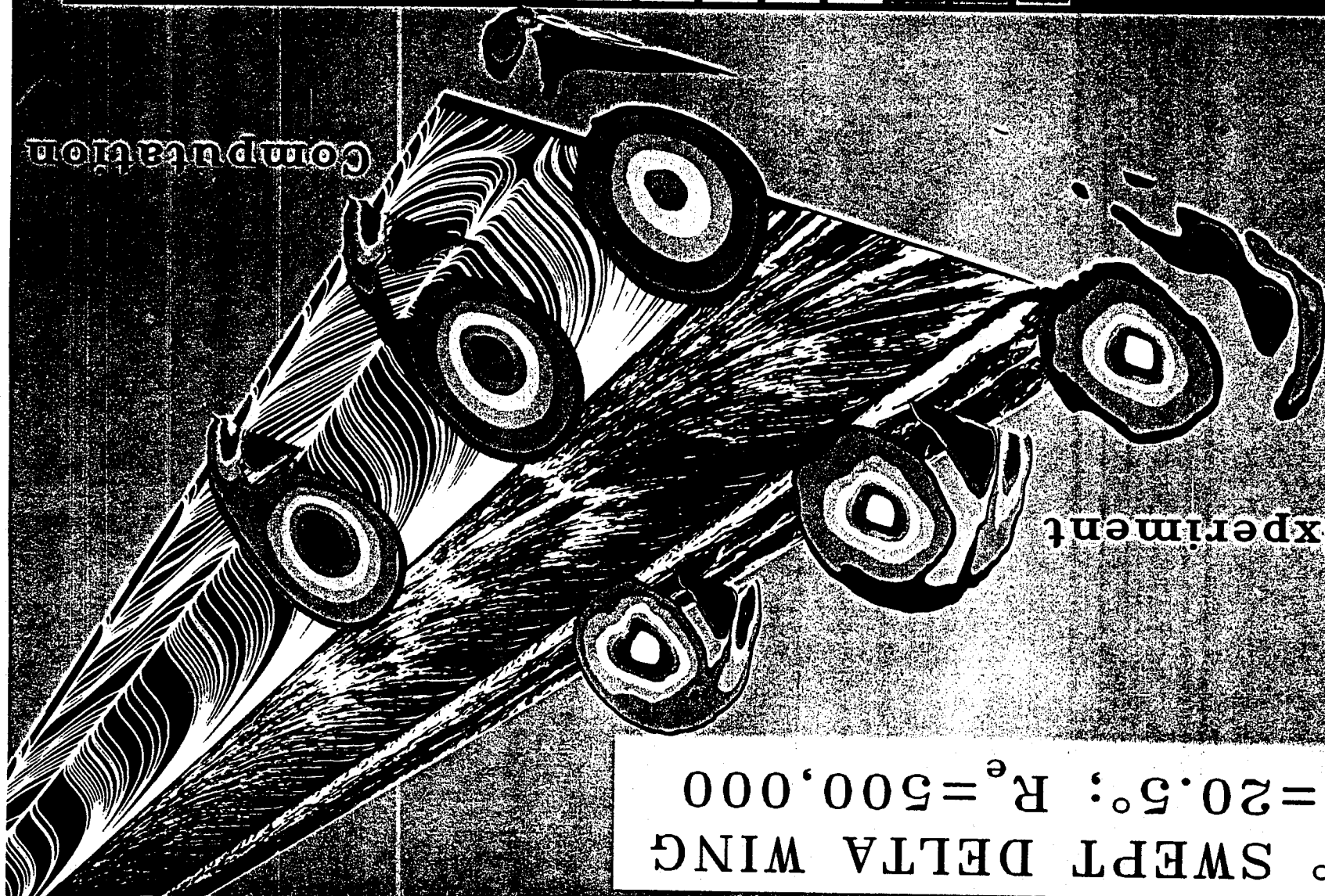
$x/L = 0.9$; $Re = 1.0$ million

Data obtained to evaluate measurement errors of 5-hole probe

75° SWEEP DELTA WING
 $\alpha = 20.5^\circ$; $R_e = 500,000$

Experiment

computation



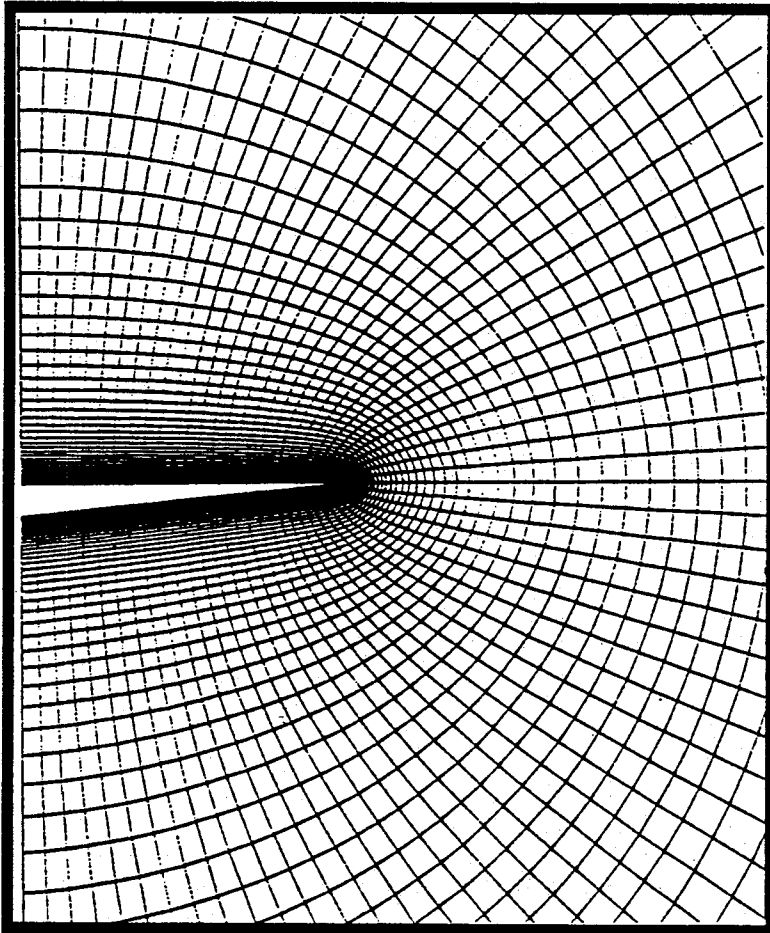
-50 0 50 100

Streamwise Vorticity

COMPUTATIONAL GRID REFINEMENTS

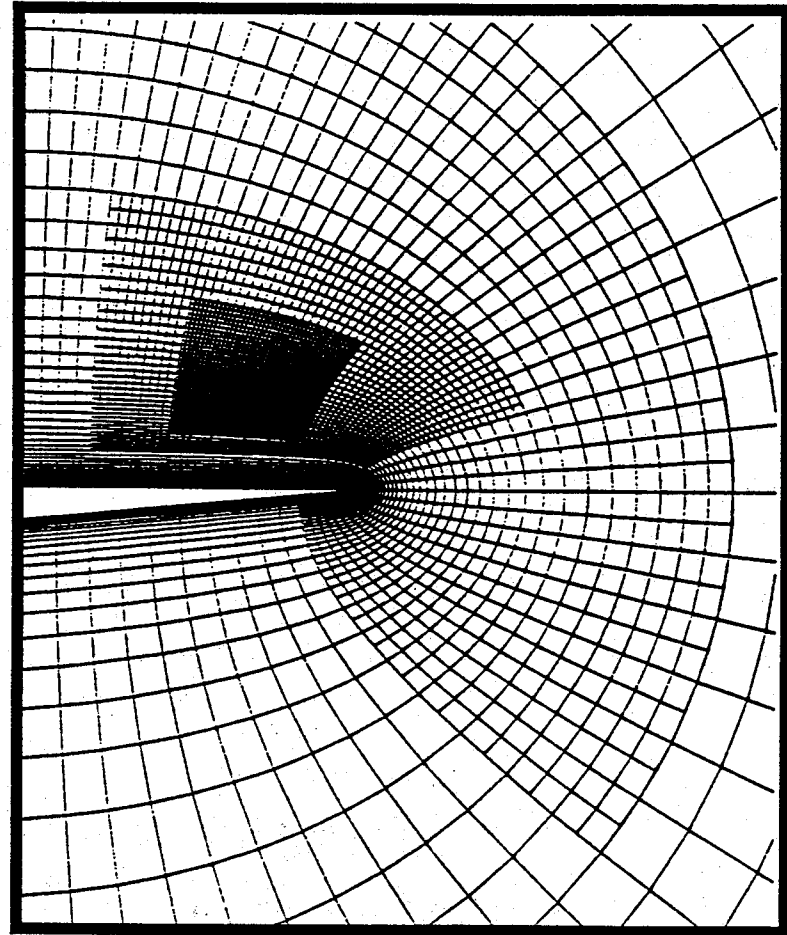
75° Delta Wing, $x/L = 0.7$

65 X 65 (4225 Total)



Single Dense Grid

4 Patches (7222 Total)

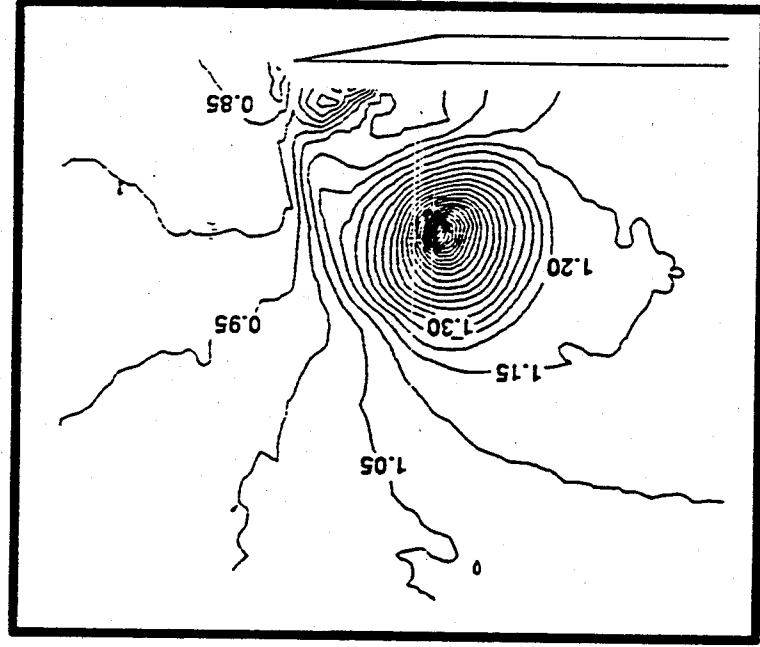


Embedded Grid

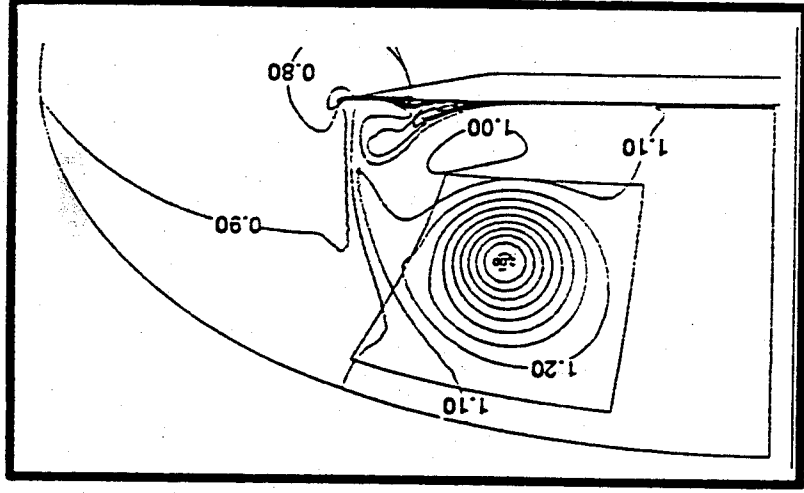
STREAMWISE VELOCITY CONTOURS

$$Re_L = 500,000; \alpha = 20.5^\circ; x/L = 0.7$$

Experiment



2 Level Refinement



Concluding Remarks

LaRC is dedicated to the development of a strong CFD Code Validation Program

- Ongoing experimental program documenting various fluid dynamic phenomena
- Experiments designed with experimental/computational cooperation
 - assessment of measurement errors
 - redundant measurements with more than one instrument, in more than one facility
 - documented results with archival storage of data for easy accessibility
- Acquisition of the required instrumentation for CFD code validation facilities
- Development of new instrumentation systems when required

52-34
116672
P25

OVERVIEW OF CFD FLOW MODELING AND VALIDATION
AMES RESEARCH CENTER

J. G. Marvin

Second NASA CFD Validation Workshop
Lewis Research Center
July 10-12, 1990

N93-70577

OUTLINE

Background

Experiments

Highlights

Instrumentation

Experiments

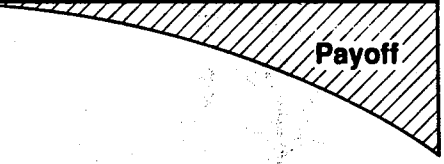
Appendix

Summaries And Key

Results of Experiments

STAGES OF CODE DEVELOPMENT AND CORRESPONDING EXPERIMENTS RELATED TO PAYOFF IN AERONAUTICAL APPLICATIONS

Science discipline	Enabling technologies	Research codes	Pilot codes	Production codes
CFD	<ul style="list-style-type: none"> • Algorithms • Grid generation • Computer Power 	<ul style="list-style-type: none"> • Technology integration • Limited pioneering applications 	<ul style="list-style-type: none"> • Wide range of applications 	<ul style="list-style-type: none"> • Applied in design environment • Cost effective
EFD	<ul style="list-style-type: none"> • Facilities • Instrumentation • Data acquisition 	<ul style="list-style-type: none"> • Building block experiments • Flow physics including numerical simulations • Flow modeling 	<ul style="list-style-type: none"> • Benchmark experiments • Calibration • Validation 	<ul style="list-style-type: none"> • Design experiments • Configuration • Performance • System integration



Payoff

Marvin -

KEY PROGRAM ELEMENTS FOR FLOW MODELING AND CFD VALIDATION AMES RESEARCH CENTER

Focus On 3-D Reynolds Averaged Navier Stokes Code Validation

Focus on Important CFD Applications

Involve CFD'ers, Modelers And Experimentalists

**Design Experiments To Provide The Specific Data Necessary To Guide
And Verify CFD Computations**

FACILITY AND INSTRUMENTATION ENHANCEMENTS 1986-PRESENT

FACILITIES REACTIVATED FOR HYPERSONICS

3.5' Hypersonic Wind Tunnel

Ballistic Range

Combustion Driven Shock Tunnel

Electric Arc Shock Tube

INSTRUMENTATION ADVANCES/DEVELOPMENTS

New Near-Wall measurement capability using LDV

New LDV Systems for Hi Re Channel And 3.5' HWT

Pressure Sensitive Chemical Luminescence Paints

LIF For Temperature And density Measurements in Wind Tunnels

Holographic System For Ballistic Range Flow Field Density Measurements

LIF For Combustion Driven Shock Tunnel-Gas Species, Tempertures

FLOW MODELING AND CFD VALIDATION EXPERIMENTS 1990

BUILDING BLOCK EXPERIMENTS

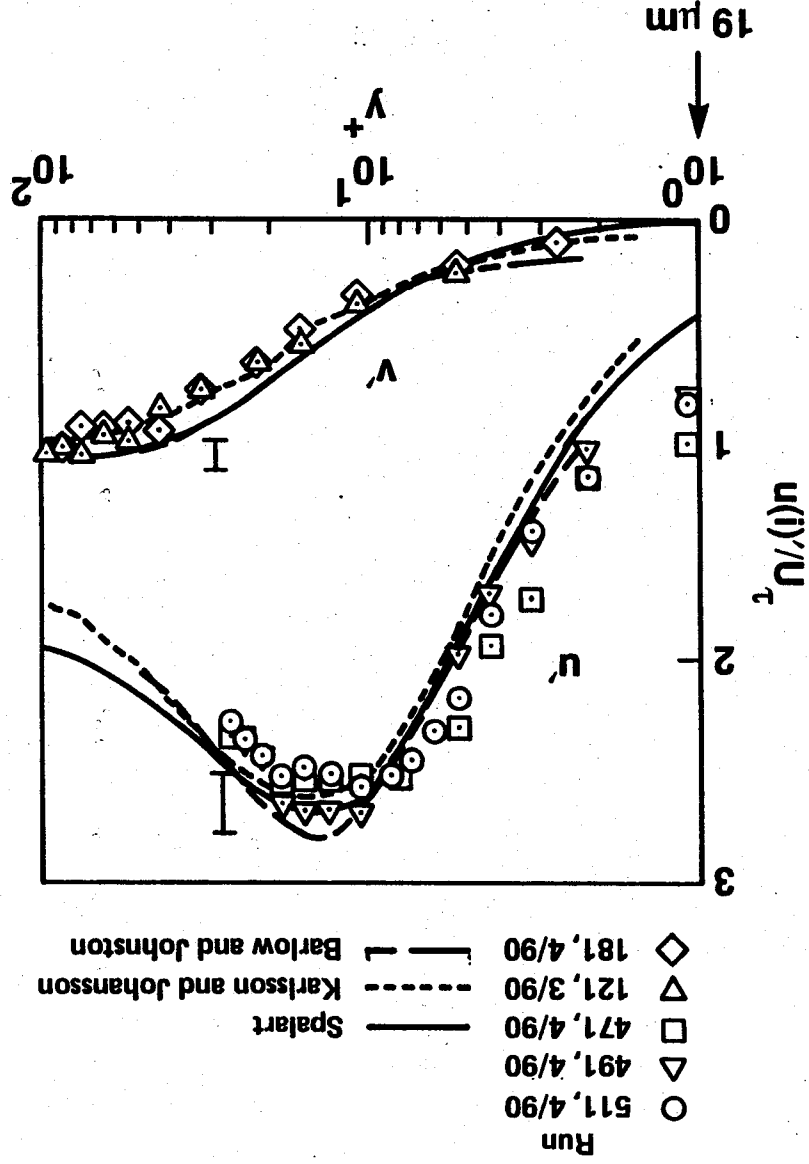
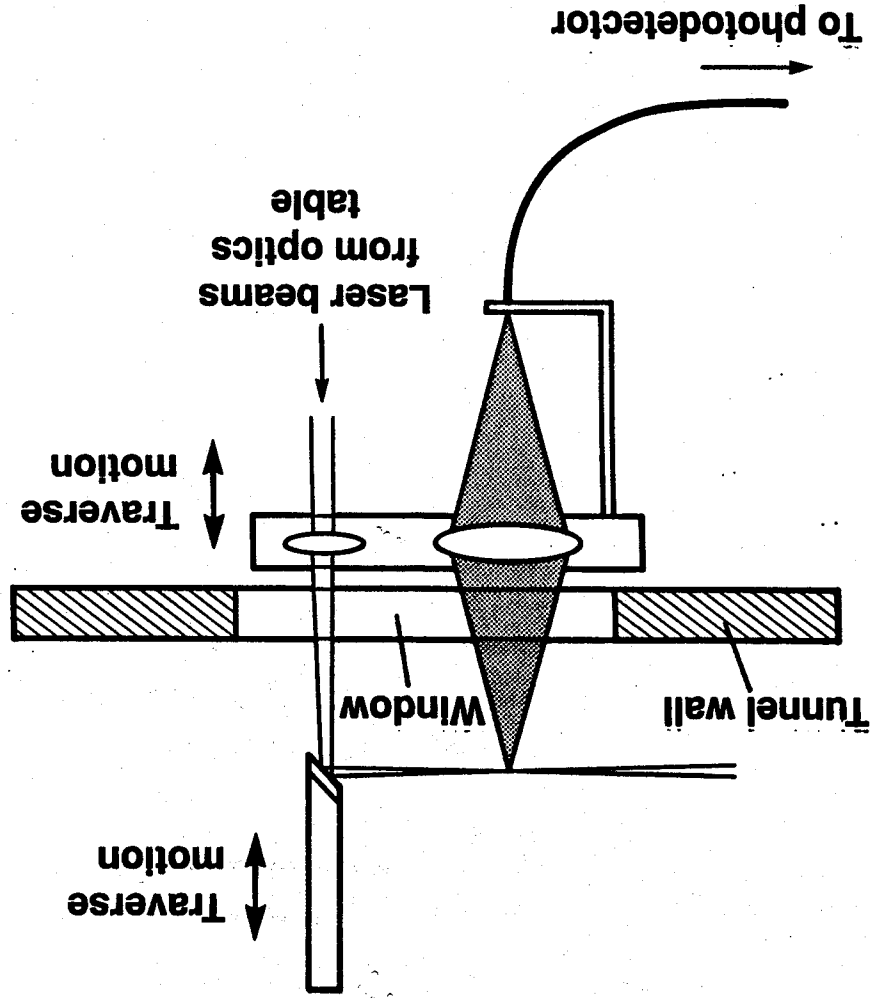
BENCHMARK EXPERIMENTS

EXPERIMENT	CONTACT	ISSUE	EXPERIMENT	CONTACT	CODE
1) Adv. Press. Grad Boundary Layer	Watmuff	Flow Physics	12) Low Aspect Ratio Wing	Olsen	TNS
2) Rearward Step	Jovic	Flow Physics	13) Airfoil w/wo Pas-sive Shock Control	Mateer	TURF
3) 3-d Spin Flow	Driver	Turb. Model	14) Generic All Body*	Lockman	UPNS
4) 3-d Wedge Flow	Johnson	Turb. Model	15) Ballistic Range-Cone Real Gas	Strawa	TUFF/STUFF
5) Wing Tip Vortex	Zilliac	Turb. Model	16) NASP Nozzle	Lockman	F3D
6) 3-d supersonic Shock Interaction	Horstman	Turb. Model	17) STOVL	Van Dalsem	F3D
7) Supersonic Shock Vortex Interaction	Settles (Penn St.)	Flow Model	18) Transonic WT Wall Boundary Conditions	Roberts (Stanford)	TNS
8) SSME Turn-Around Duct*	Monson	Turb. Model			
9) Coanda	Brown	Turb. Model			
10) Hypersonic Shock Interaction	Horstman	Turb. Model			
11) Shock Tube	Sharma	Reaction Rates			

* Chosen For Technical Presentation

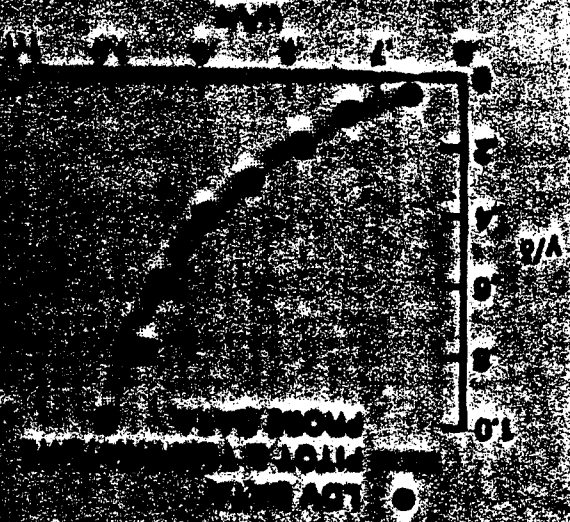
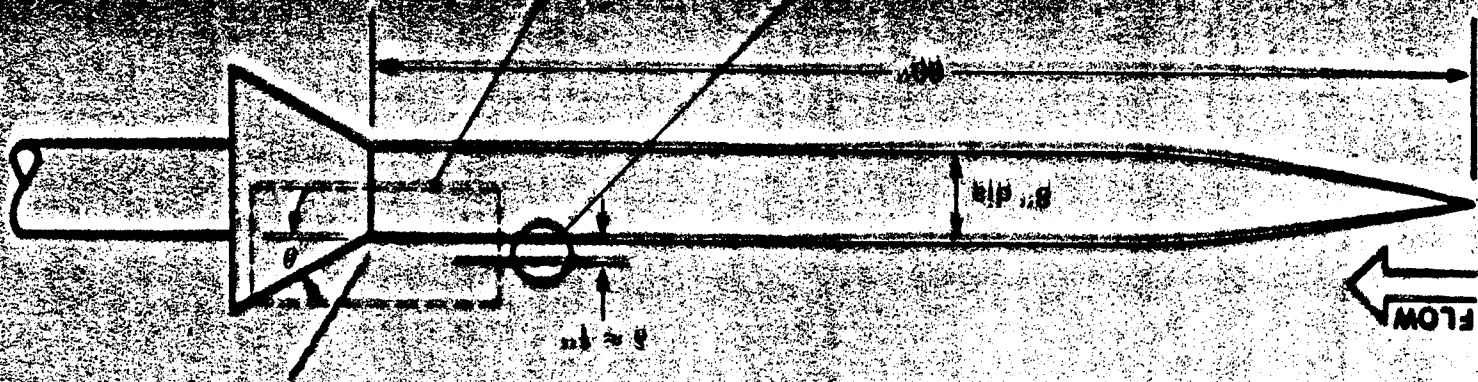
KEY ACCOMPLISHMENTS
← **Instrumentation**
Experiments

LASER DOPPLER VELOCIMETER FOR 3-D NEAR-WALL MEASUREMENTS



LDV FEASIBILITY STUDY IN A HYPERSONIC SHOCK-WAVE/ TURBULENT-BOUNDARY-LAYER INTERACTION

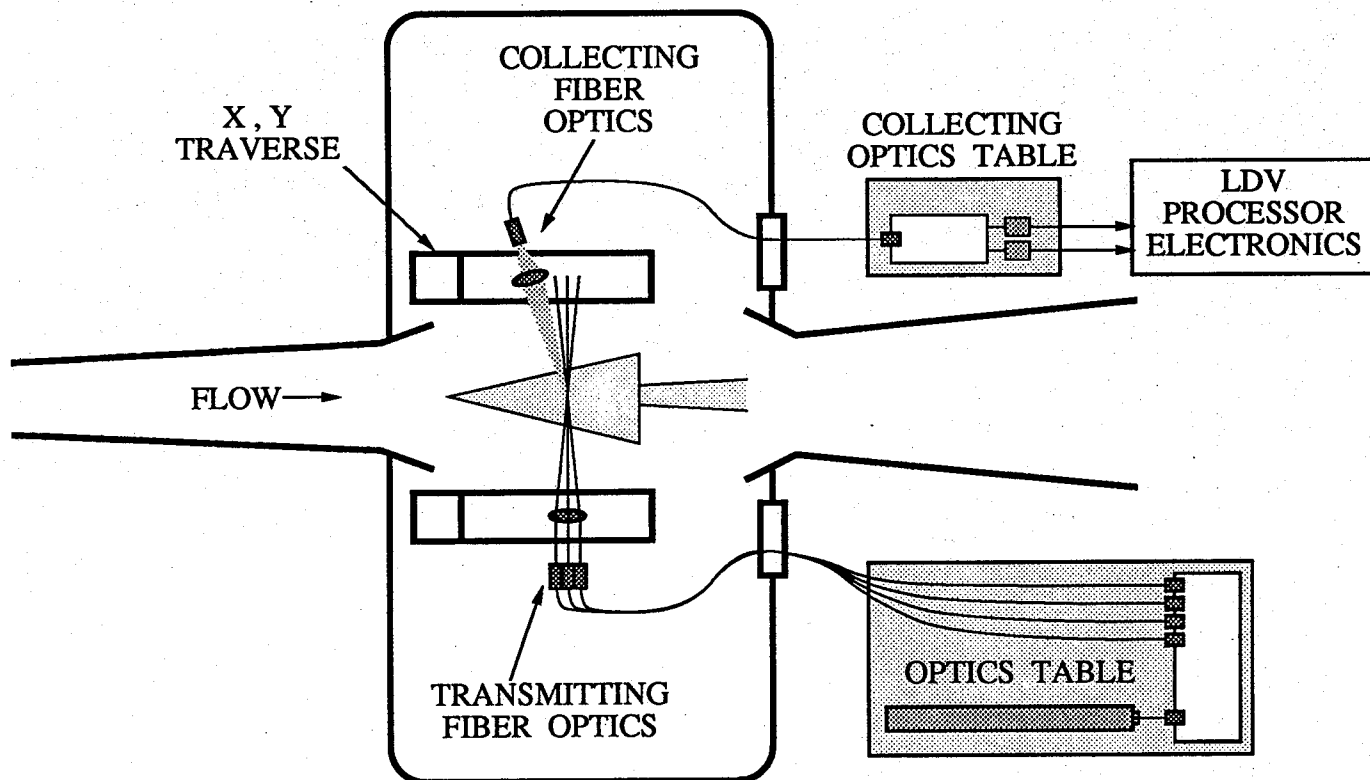
$M_\infty = 7$ $Re_{x_{tr}} = 18 \times 10^6$



ORIGINAL PAGE IS
OF POOR QUALITY

OF 1700 0000
1000 1000 1000

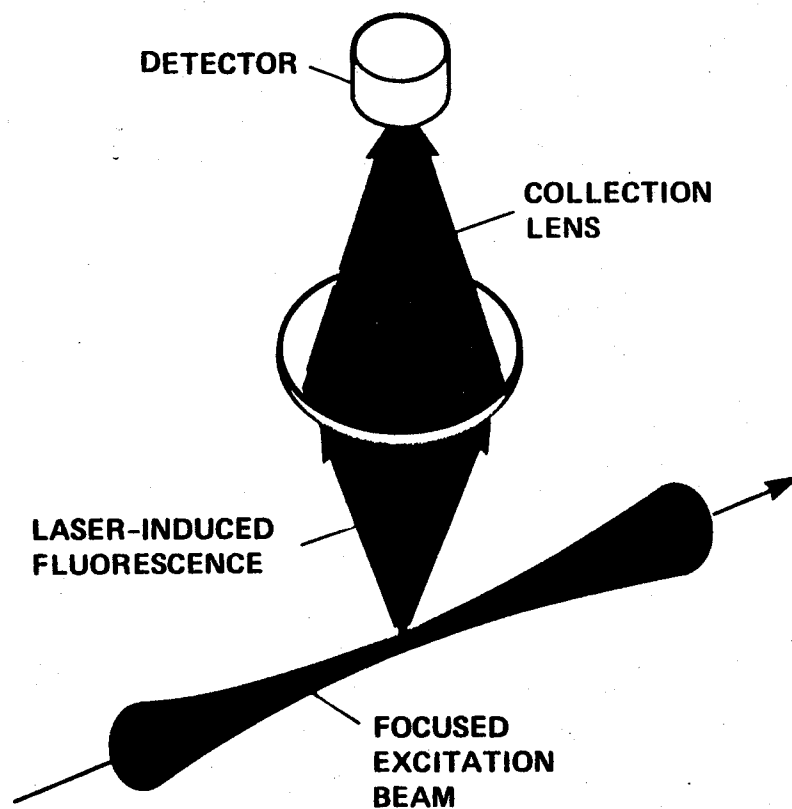
3.5 FOOT HWT LDV SYSTEM



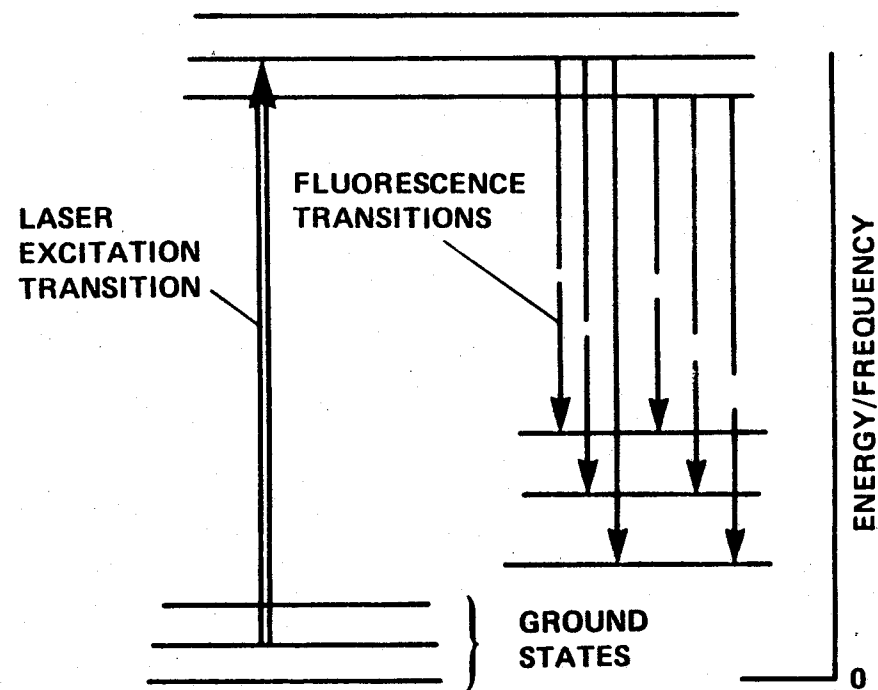
COMPLERE INC.

LASER-INDUCED FLUORESCENCE (LIF)

MEASURES TEMPERATURE, DENSITY, SPECIES

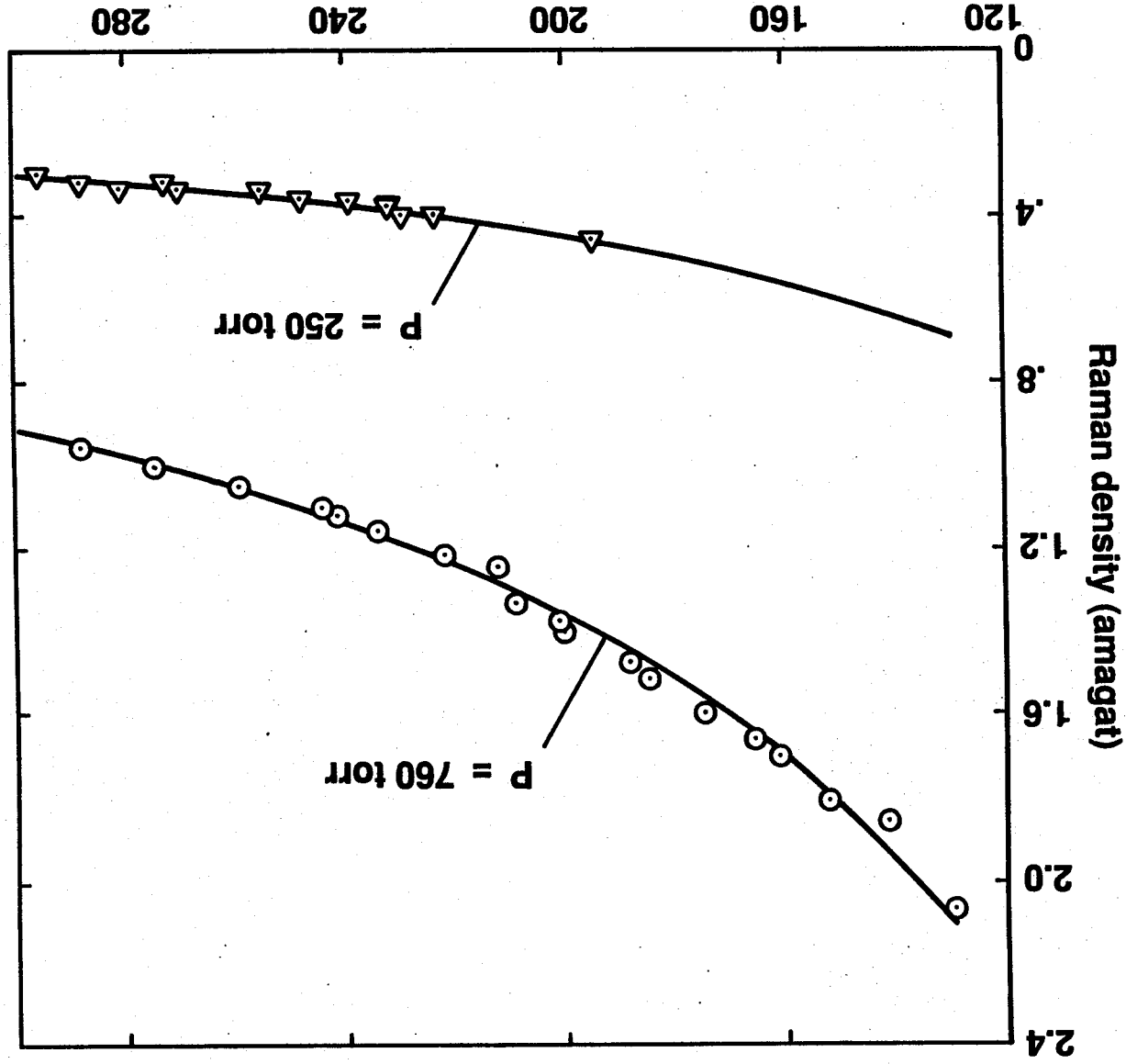


**EXPERIMENTAL
CONFIGURATION**



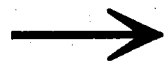
**MOLECULAR ENERGY LEVELS
AND RADIATIVE FREQUENCIES**

TEMPERATURE AND DENSITY MEASUREMENTS IN AIR AT CONSTANT PRESSURE USING LIF/RAMAN SPECTROSCOPY



KEY ACCOMPLISHMENTS

Instrumentation



Experiments

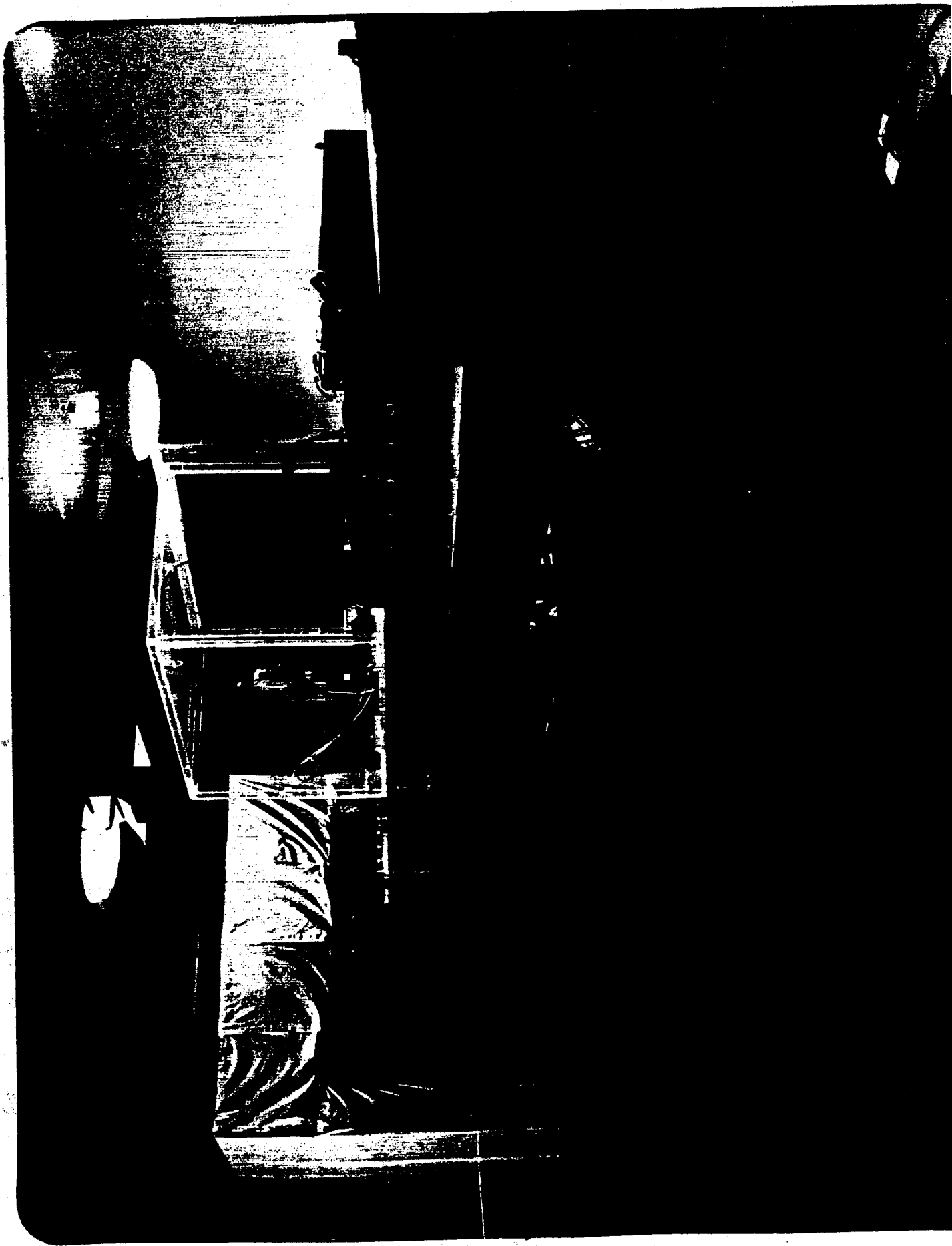
1/15/1

NAME

NO. AC80015-4

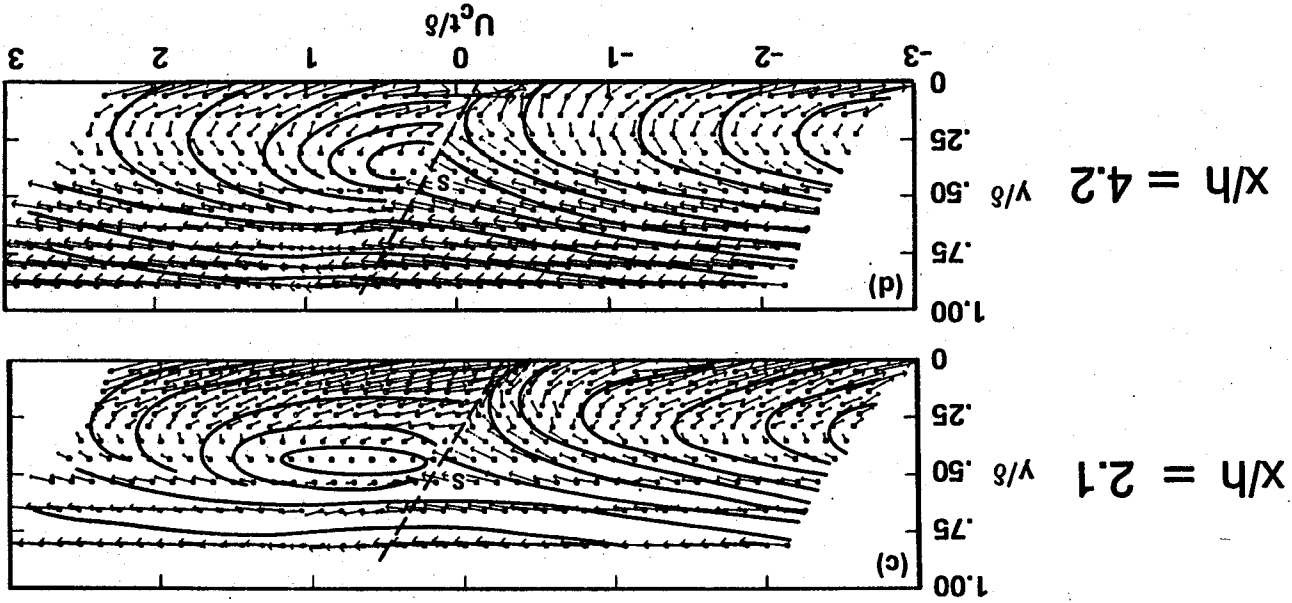
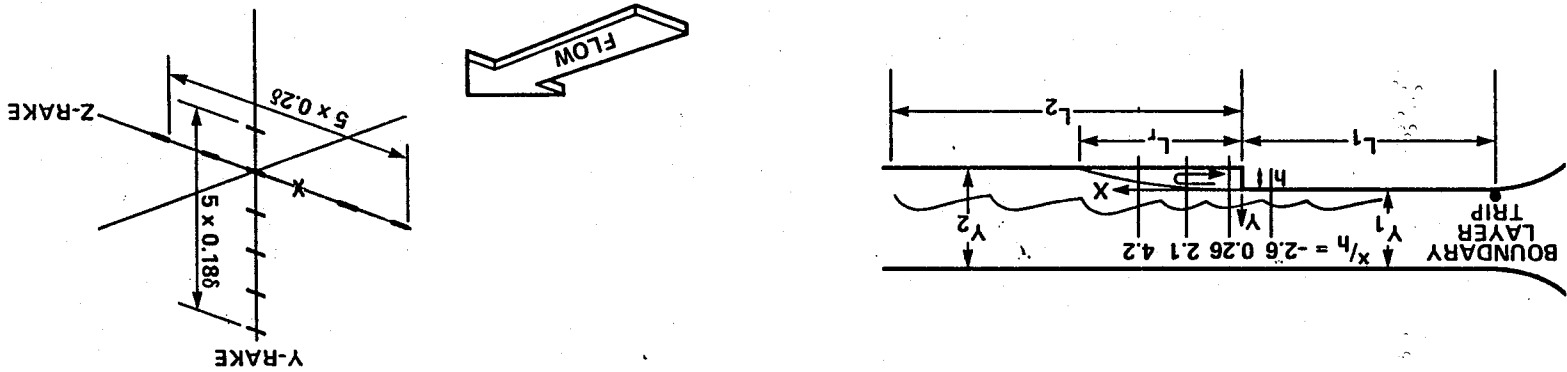
ORIGINAL PAGE IS
OF POOR QUALITY

MARVIN - 13



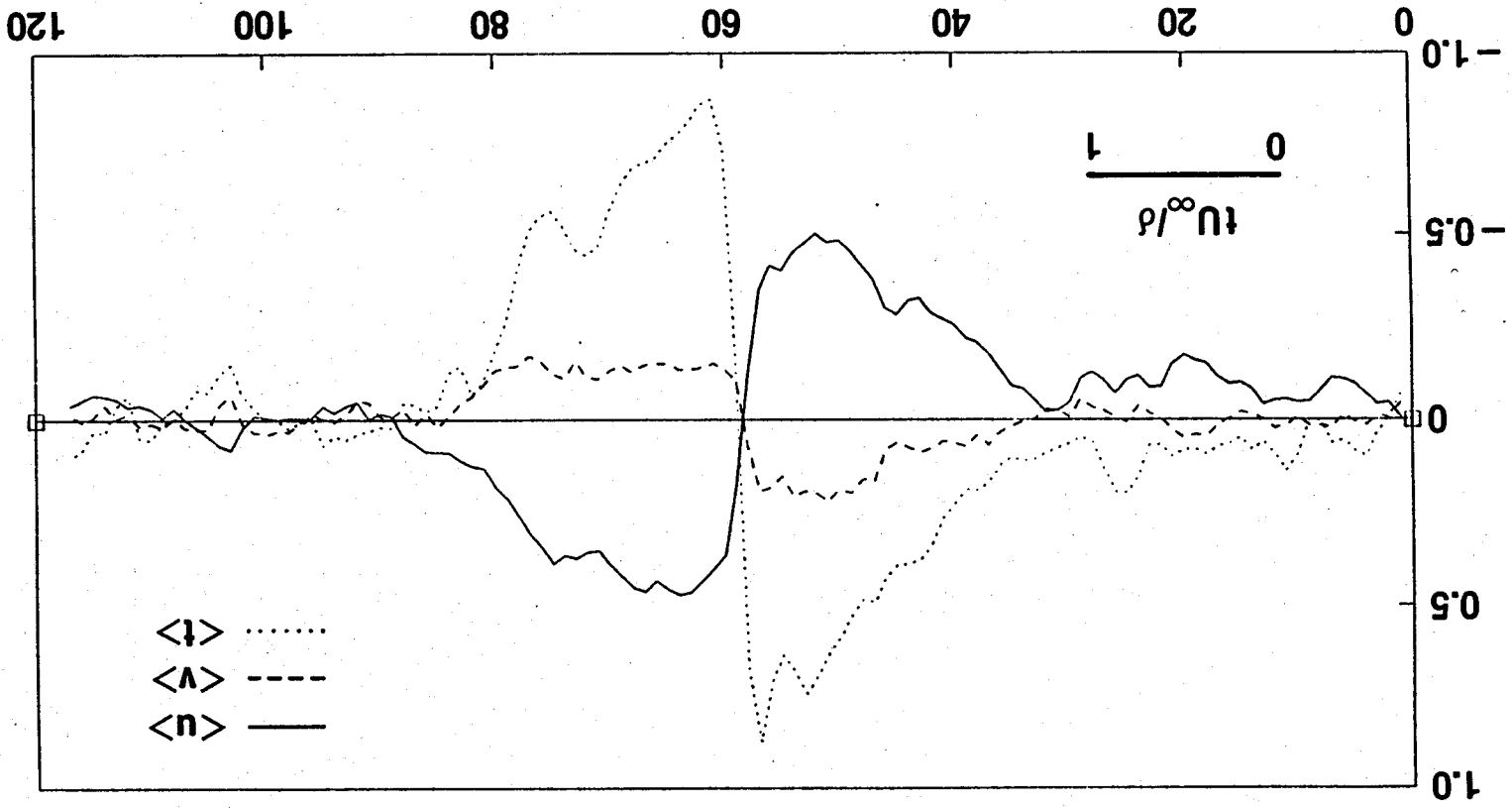
TOPOLOGY OF COHERENT STRUCTURES

$Re_\theta = 2000 \quad U_\infty = 10 \text{ m/sec}$



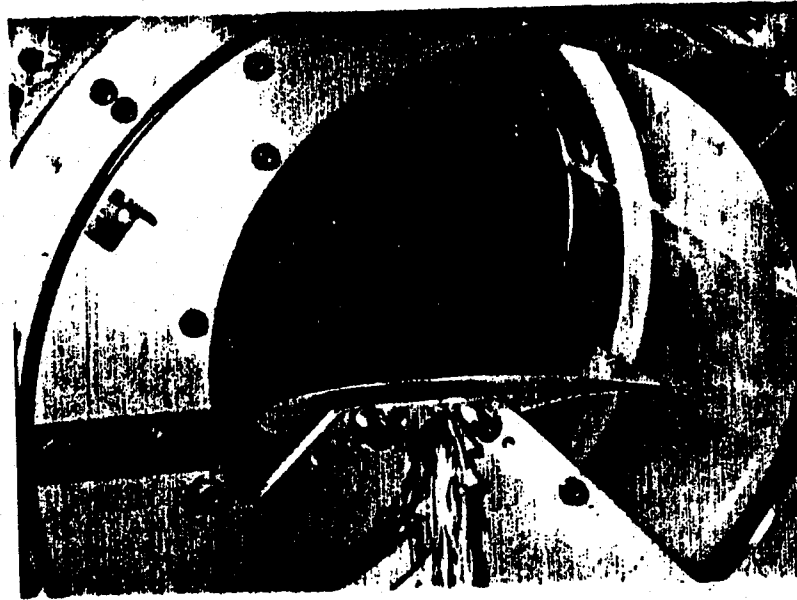
ENSEMBLE AVERAGED QUANTITIES

$$X/h = -2.6 \quad y^+ = 260$$



VERIFICATION EXPERIMENT FOR TRANSONIC FLOW CODES

SUPERCritical AIRFOIL WITH LDV



TEST CONDITIONS

$$0.73 < M < 0.80$$

$$2 \times 10^6 < Re < 6 \times 10^6$$

$$0.5 < \alpha < 1.5$$

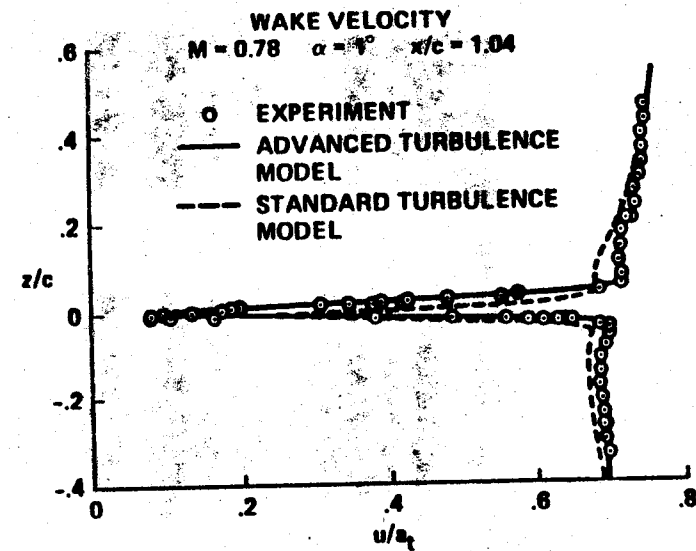
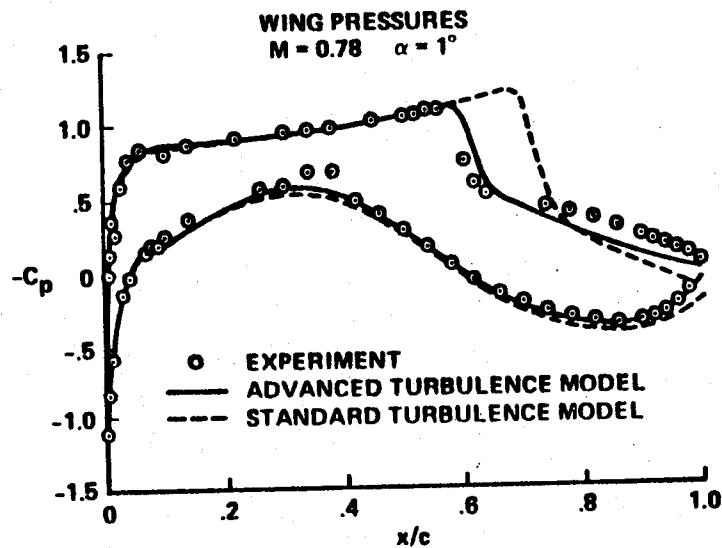
MEASUREMENTS

PRESSURE (\bar{p} , P^+)

FLOW FIELD (U , V , $\overline{u'v'}$...)

OIL FLOW

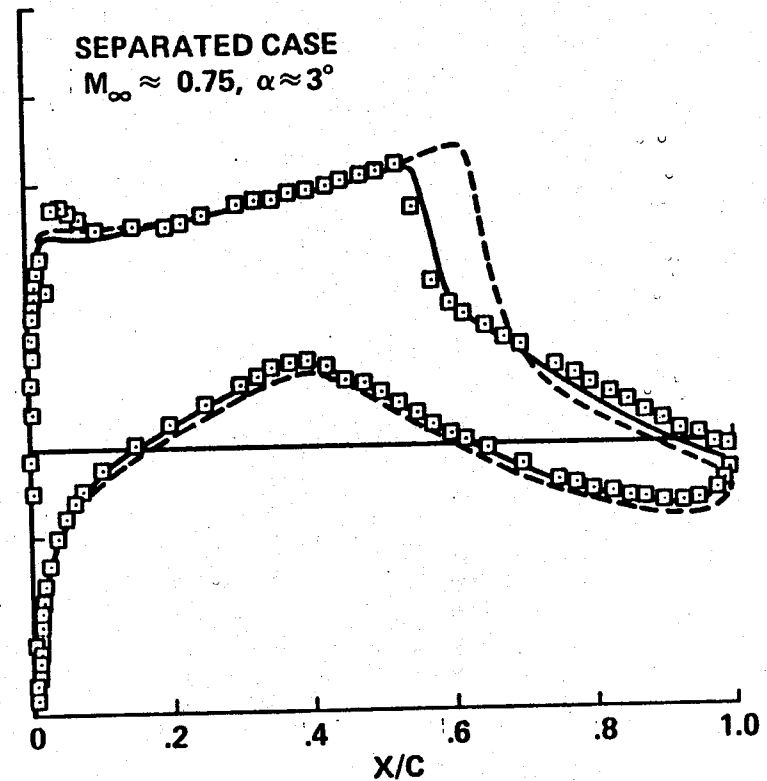
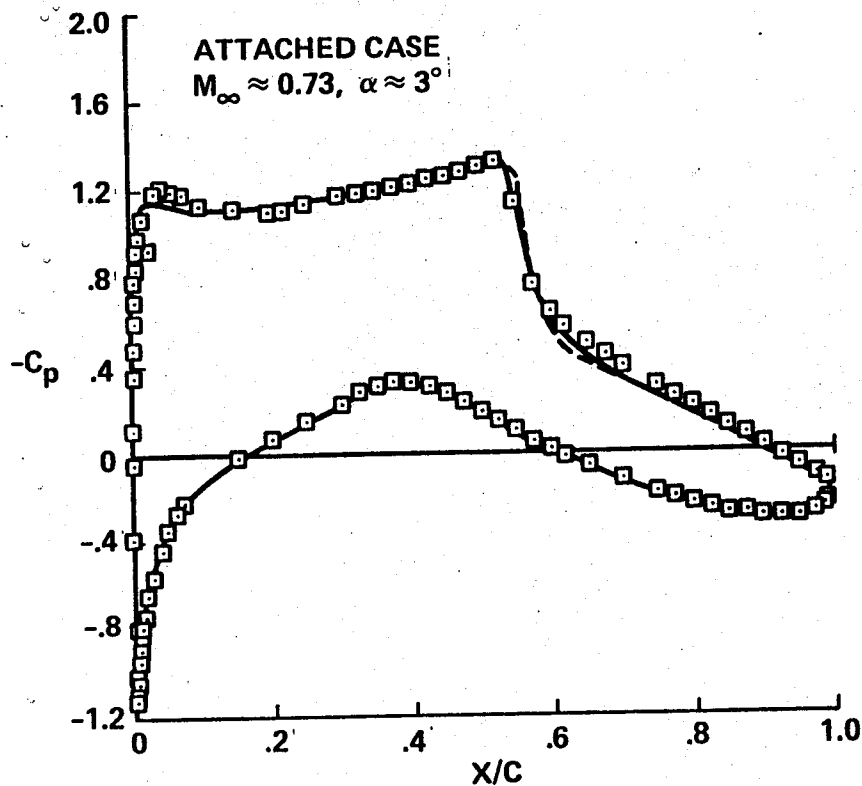
LIFT, DRAG



RANS TRANSONIC AIRFOIL PREDICTIONS

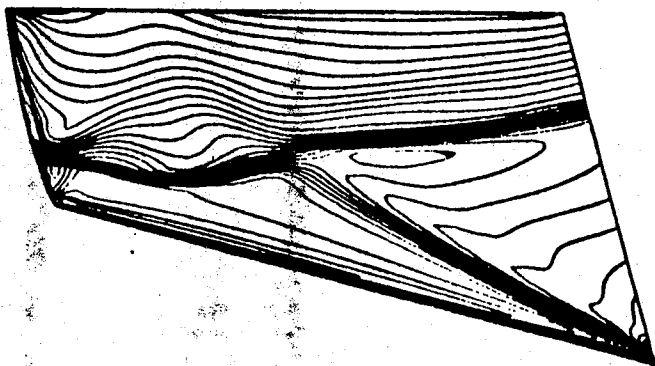
TURBULENCE MODELING IMPROVEMENTS RAE 2822 AIRFOIL

□ EXPERIMENT
— NONEQUILIBRIUM MODEL
--- EQUILIBRIUM MODEL

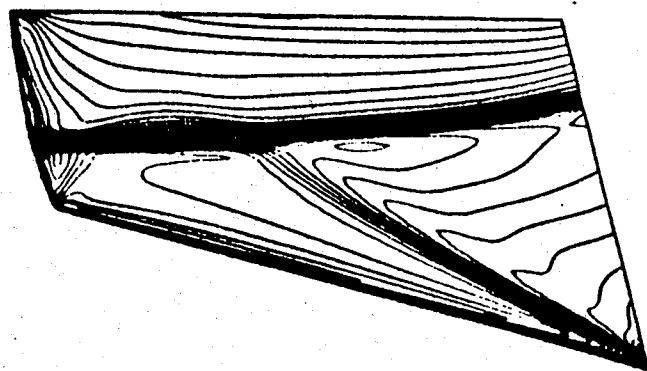
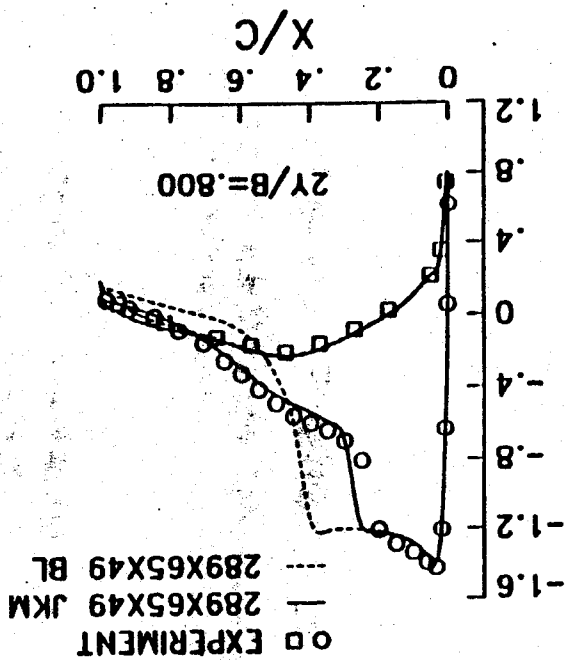


EFFECT OF TURBULENCE MODEL ON WING PRESSURES ONERA M-6 WING, $M=0.84$ $\alpha = 6^\circ$

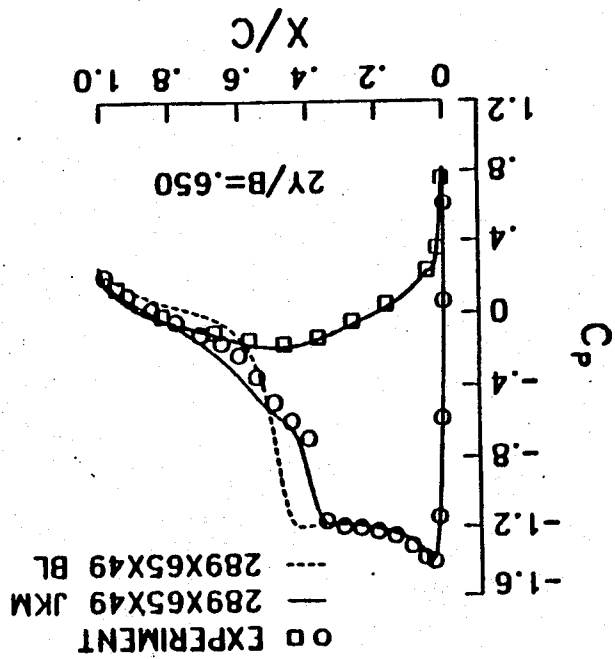
PRESSURE CONTOURS



JOHNSON-KING



BALDWIN-LOMAX

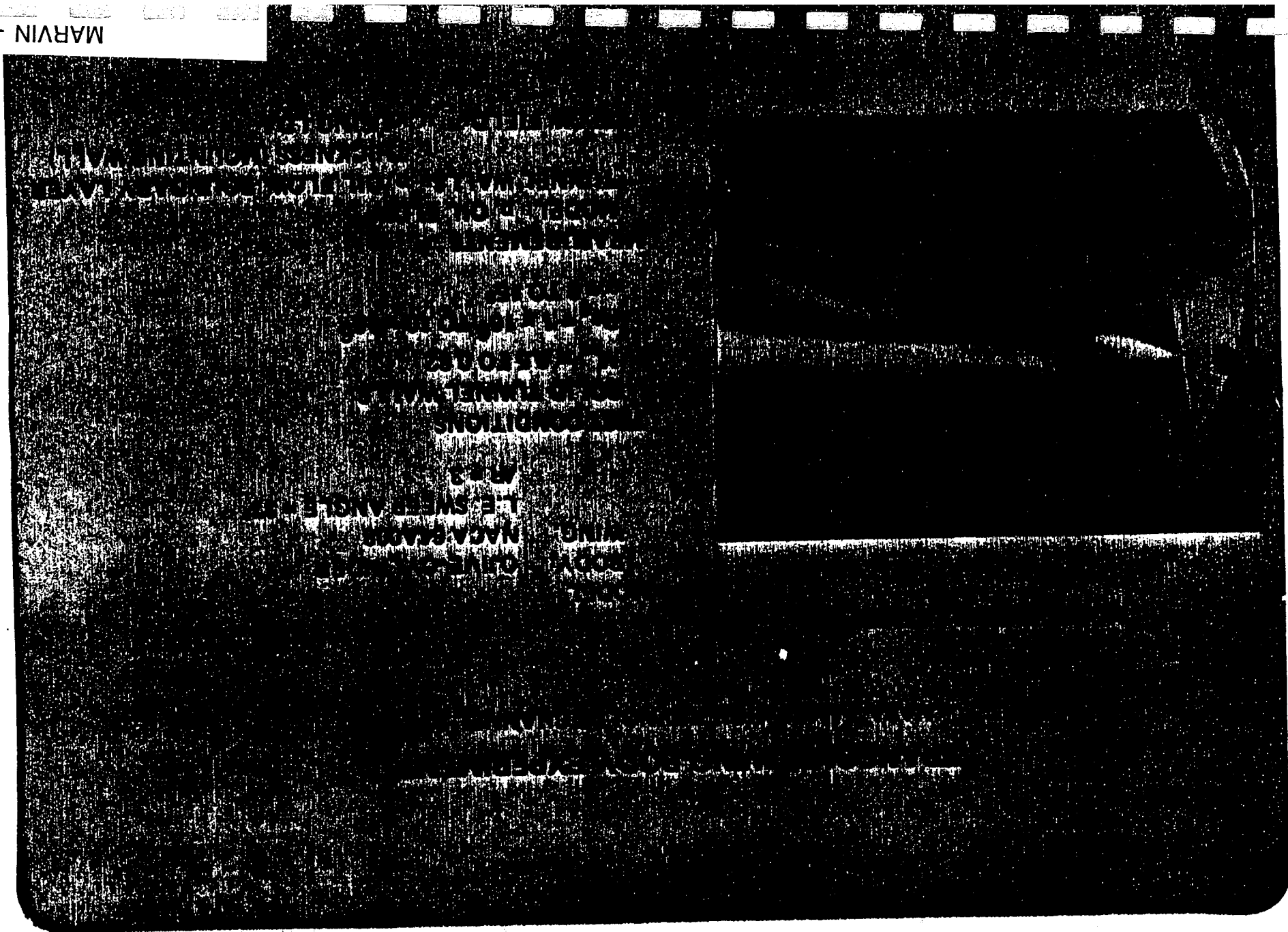


NAME _____

NO. 487-0704-1.1

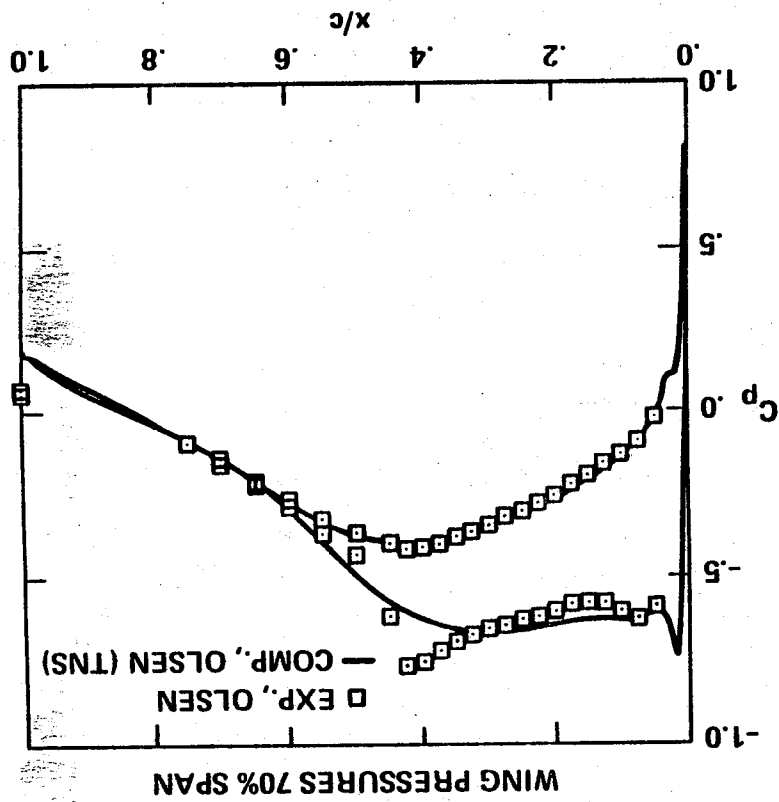
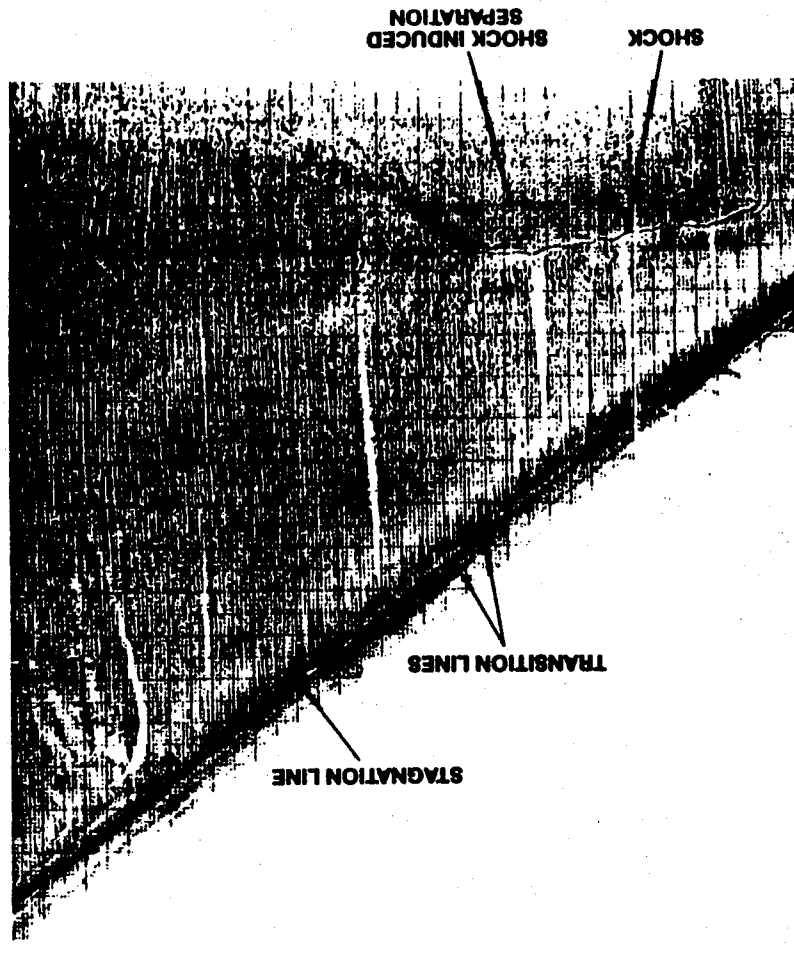
ORIGINAL PAGE IS
 OF POOR QUALITY

MARVIN - 19



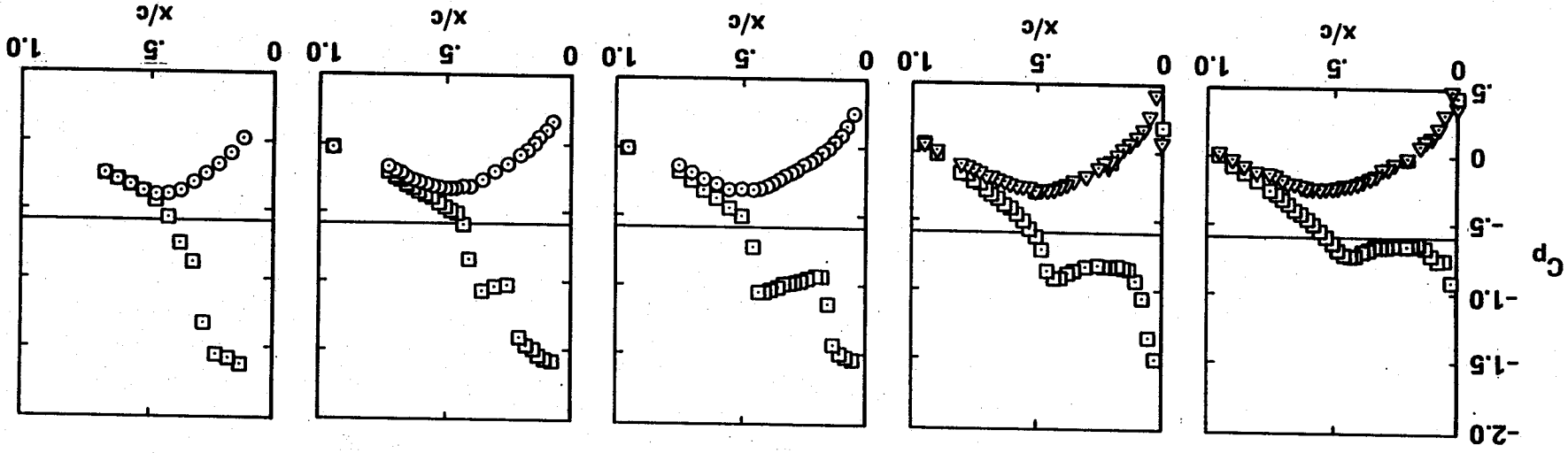
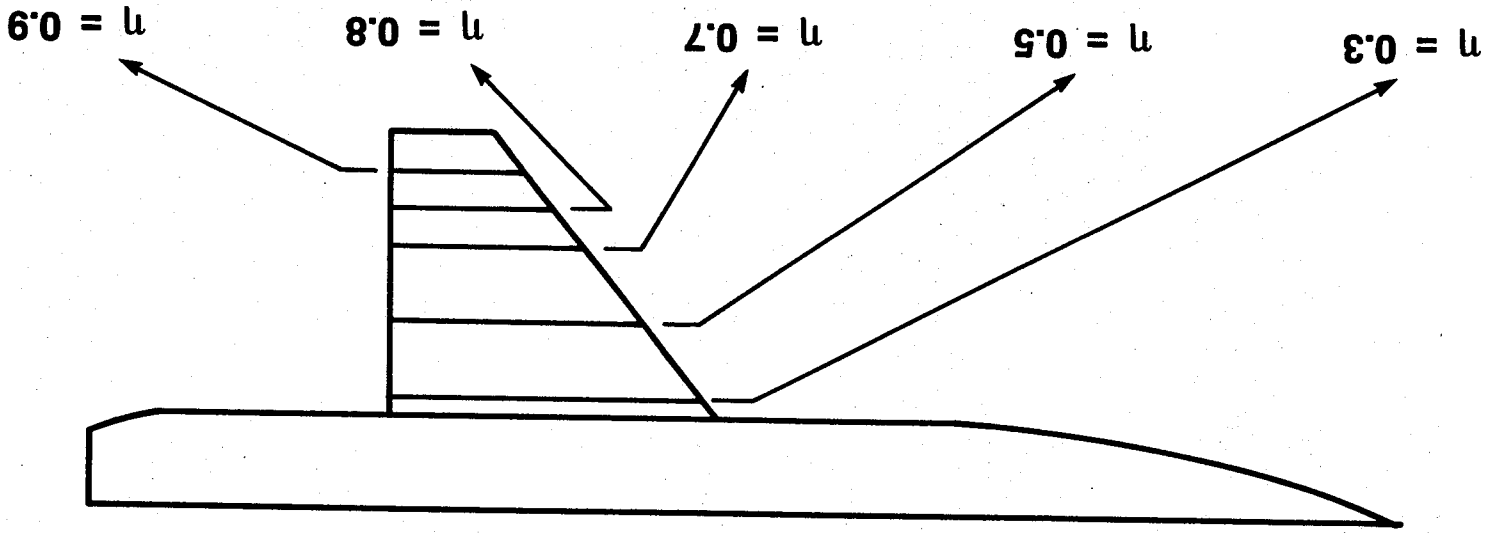
TRANSONIC WING-BODY EXPERIMENT

WING-ALONE RESULTS
 $M_\infty = 0.8$, $Re_c = 9 \times 10^6$



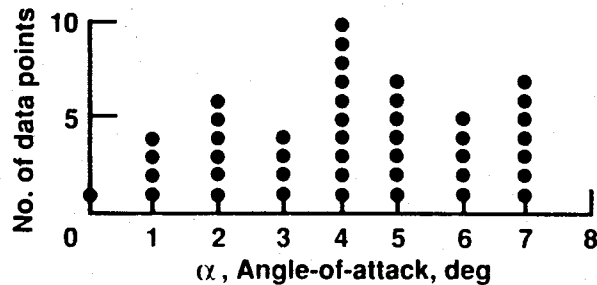
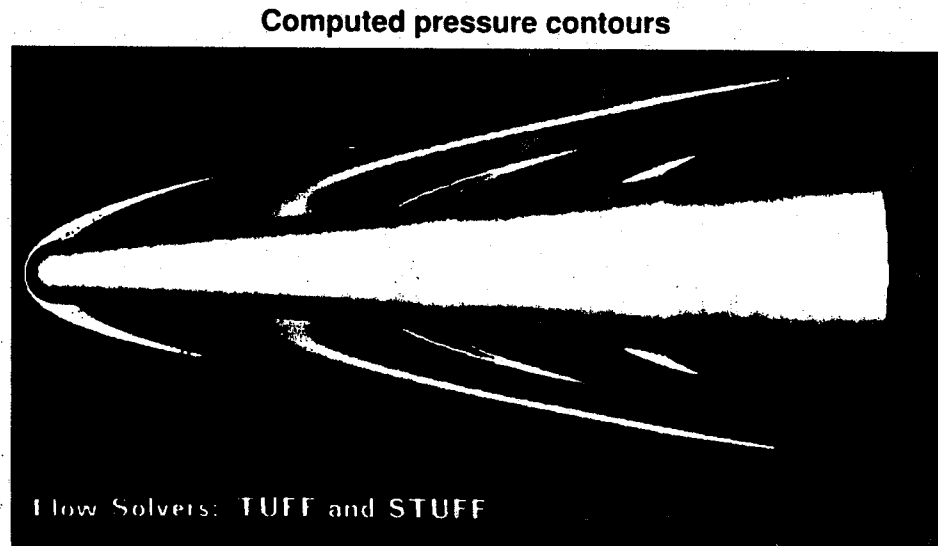
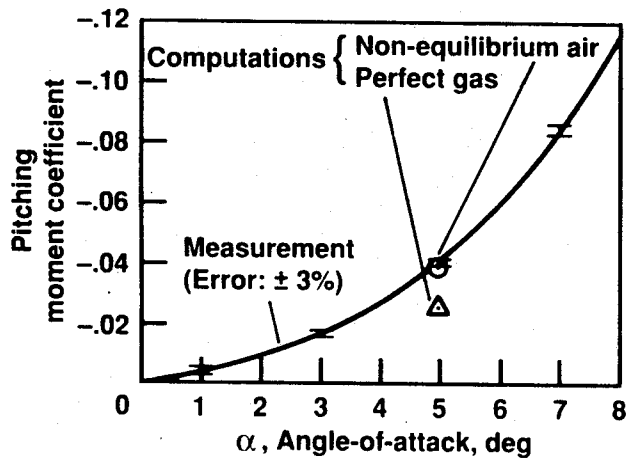
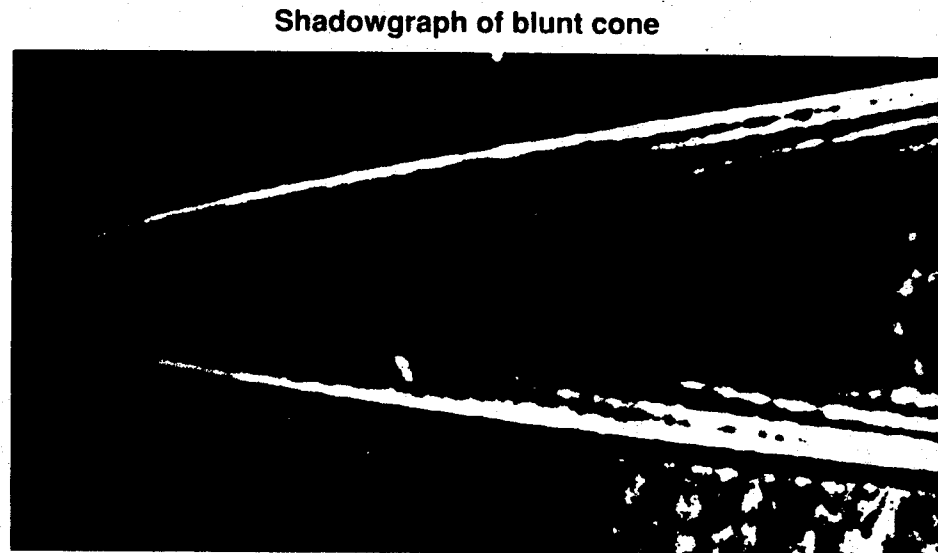
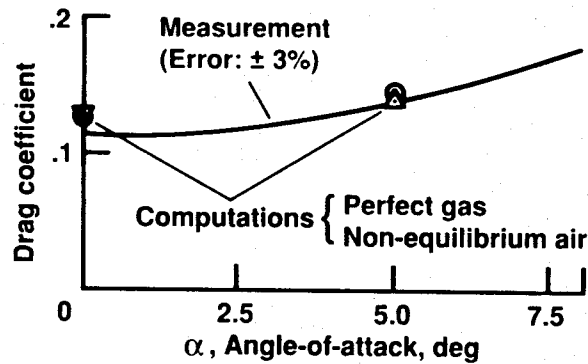
LOW ASPECT RATIO WING/BODY EXPERIMENT

$Re_{Cr} = 12 \times 10^6$ $\alpha = 5^\circ$ $M_\infty = 0.775$



REAL-GAS FORCE AND MOMENT MEASUREMENTS AND CFD CORRELATION FROM THE BALLISTIC RANGE

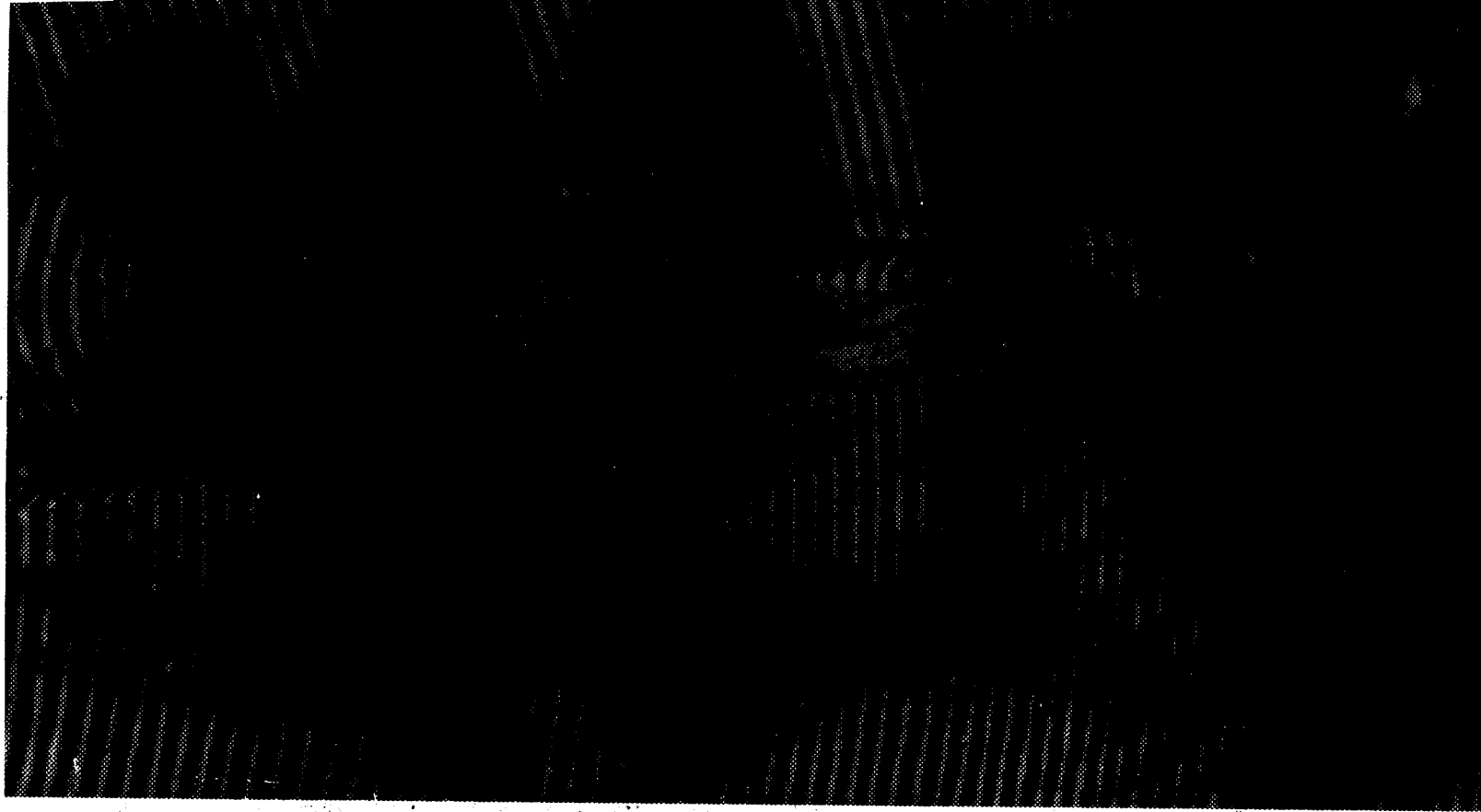
$Re = 10^6$, $V_\infty = 5$ km/sec, $Mn = 14.5$



NASA - Ames Research Center
Hypervelocity Free Flight Facility - March 1990

FINITE FRINGE INTERFEROGRAM

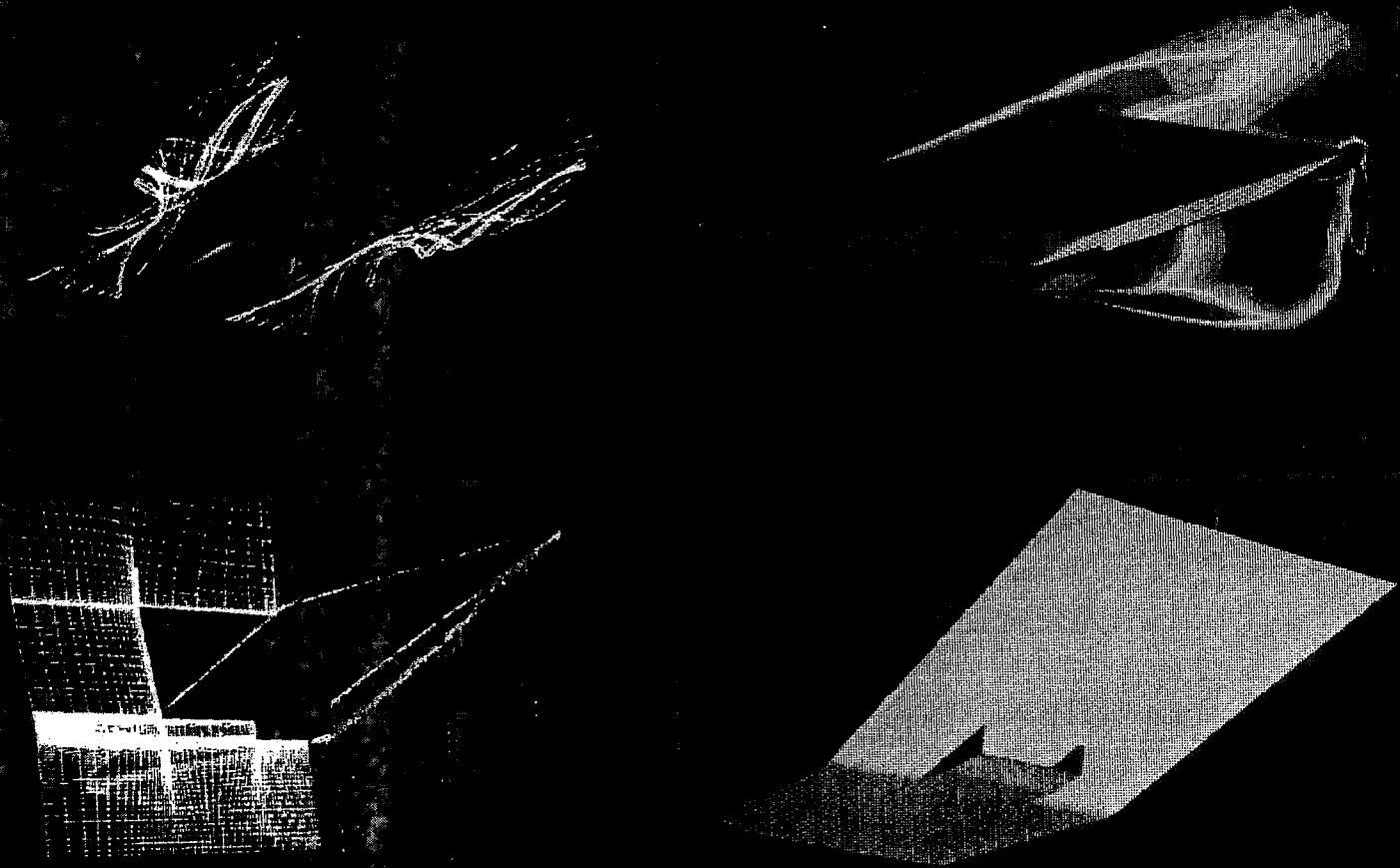
5 degree half-angle blunt cone
 $p = 63\text{mm}$
 $v = 16,400\text{ ft/sec}$



ORIGINAL PAGE IS
OF POOR QUALITY

NASP Nozzle/Afterbody Computations

Experimental Geometry
3-D Composite Grid Topology



Pressure Contours & Particle Traces

S.M. Ruffin & E. Venkatapathy

Mach Contours in Symmetry & Outflow Planes

53-34

116673

P-30

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

BY

L.A. SCHUTZENHOFER, H.V. McCONNAUGHEY, P.K. McCONNAUGHEY

COMPUTATIONAL FLUID DYNAMICS BRANCH

AEROPHYSICS DIVISION

STRUCTURES AND DYNAMICS LABORATORY

NASA/MARSHALL SPACE FLIGHT CENTER

SECOND NASA CFD VALIDATION WORKSHOP

JULY 10-12, 1990

NASA LEWIS RESEARCH CENTER

N93-70578

5-9787-0-238

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

OVERVIEW

- FOCUS OF MSFC CFD ACTIVITIES
- DESIGN APPLICATIONS
- VALIDATION ACTIVITIES
- CFD REQUIREMENTS
- CFD EXPECTATIONS
- SUMMARY

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

FOCUS OF MSFC CFD ACTIVITIES

● SUPPORT PROGRAM OFFICES

- "QUICK TURNAROUND" APPLICATIONS; SENSITIVITY ANALYSIS; RETROFITTABLE DESIGN OPTIONS
- INTERACT WITH HARDWARE CONTRACTORS IN DEVELOPMENT OF DESIGN ENVIRONMENTS
- PROVIDE "SMART BUYER" CAPABILITY FOR LONG-TERM APPLICATIONS
- DEVELOP SUBSYSTEMS CFD MODELS
- FOCUS MSFC CFD ACTIVITIES/PROVIDE CENTERWIDE CFD SUPPORT

● FOCUS DEVELOPMENT OF CFD METHODOLOGY

- INTERACT WITH ARC, LeRC, LaRC, AND OTHER RESEARCH ORGANIZATIONS TO FOCUS TECHNOLOGY DEVELOPMENT TOWARDS MSFC HARDWARE RELATED PROBLEMS
- DEVELOP REQUIREMENTS FOR CFD CODE VERIFICATION
- VERIFY CODES THROUGH BENCHMARK COMPARISONS
- ADVANCE CFD TECHNOLOGY FOR APPLICATIONS

● DEVELOP ADVANCED HARDWARE TECHNOLOGY CONCEPTS

- TURBINE STAGE
- PUMP STAGE
- NOZZLES, PREBURNERS, ETC.

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

DESIGN APPLICATIONS

- PROGRAM SUPPORTING ACCOMPLISHMENTS
- DESIGN APPLICATION EXAMPLES
 - ATD DISCHARGE VOLUTE SIDE LOAD ANALYSIS
 - MAIN INJECTOR LOX INLET FLOW INDUCED VIBRATION
 - NOZZLE/MAIN COMBUSTION CHAMBER (MCC) MISMATCH
- CURRENT PROGRAMS

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

PROGRAM SUPPORTING ACCOMPLISHMENTS

		<u>INHOUSE</u>	<u>CONT.</u>
SSME	• HPFTP TURBINE BLADES	X	
	• TURBINE DISK CAVITIES	X	
	• LOX PUMP BEARING INLET CAVITY	X	
	• LOX PUMP BEARINGS		X
	• FUEL PREBURNER	X	
	• LOX PREBURNER	X	
	• LOX MANIFOLD TEE (4000 Hz)	X	X
	• HOT GAS MANIFOLD/MANIFOLD STRUTS	X	X
	• PUMP COOLANT FLOW PATHS	X	
	• NOZZLE/MCC MISMATCH	X	X
	• HPOTP NOZZLE PLUG TRAJECTORIES	X	X
	• TRANSIENT BEHAVIOR OF FUEL PREBURNER MANIFOLD		X
	• BEARING DEFLECTOMETER (TTBE)	X	
	• ENGINE 0212 BEARING CAGE DEBRIS	X	
	• ENGINE 0209 MCC COOLANT LINER CRACKING	X	
	• ENGINE 2019 HPFTP STATIC SEAL PARTICLE TRAJECTORIES	X	
ATD	• TURBINE INLET TEMP. REDISTRIBUTION	X	X
	• TURBINE TEMP. PROFILE REDISTRIBUTION		X
	• ROTOR-STATOR INTERACTION		X
	• TURNAROUND DUCT AND HOT GAS MANIFOLD		X
	• BEARING ANALYSIS		X
	• LOX PUMP INLET SCROLL		X
	• FUEL PUMP INTERSTAGE CROSSOVER DUCTS		X
	• FUEL PUMP INLET SCROLL		X
	• LOX PUMP DISCHARGE VOLUTE	X	X
	• SEALS		X
	• HOT GAS MANIFOLD CHECKOUT CHAMBER	X	
	• FUEL PUMP DISCHARGE VOLUTE	X	
	• HPFTP AFT DISK CAVITY	X	
	• HPOTP TURNAROUND DUCT VANE FAILURE	X	
	• UNSTEADY MULTISTAGE TURBINE LOADS	X	

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

PROGRAM SUPPORTING ACCOMPLISHMENTS

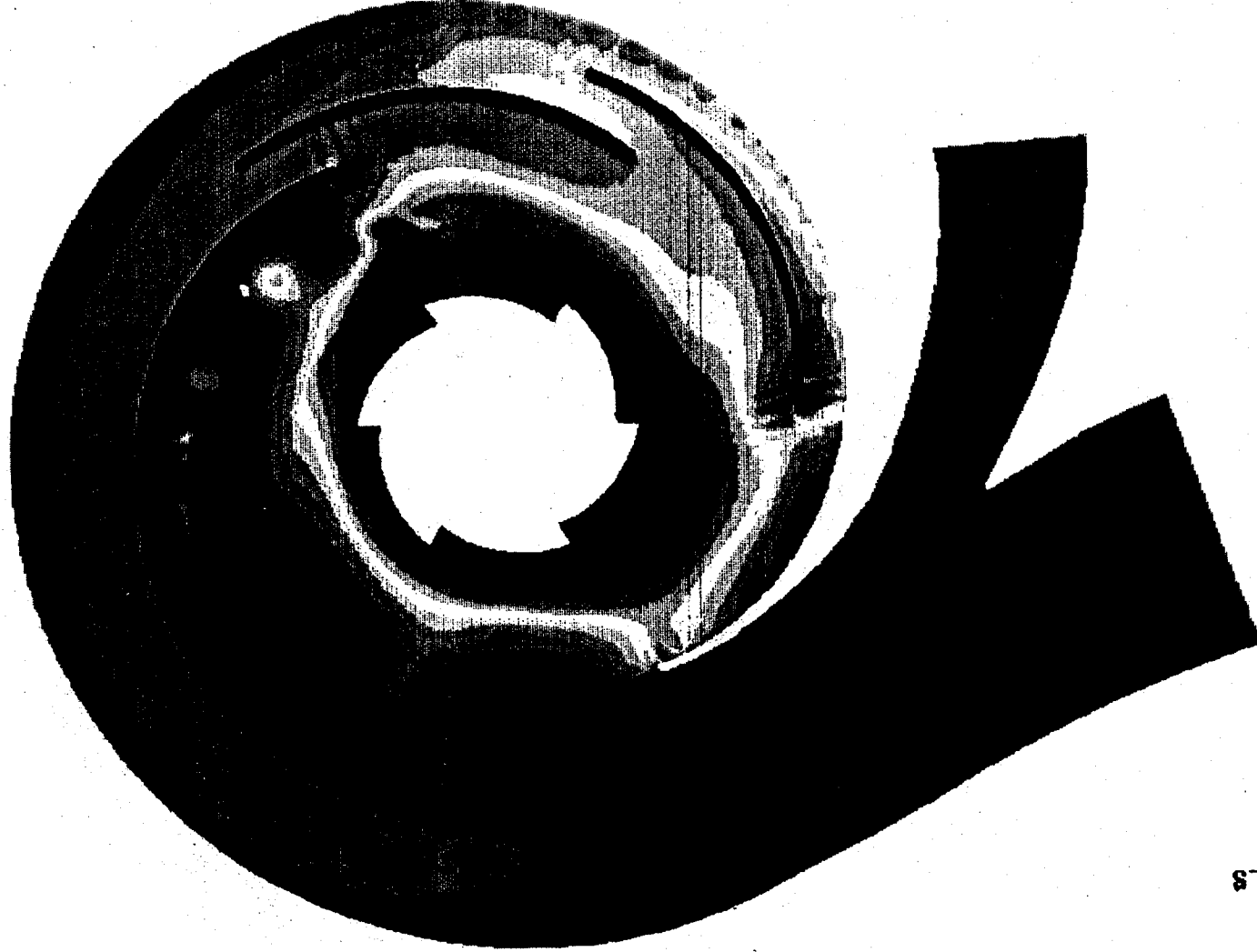
		<u>INHOUSE</u>	<u>CONT.</u>
SRB	● BORE FLOW		
	— CANTED NOZZLE	X	X
	— BROKEN INHIBITOR		X
	● FIELD JOINT		
	— FLOW AND THERMAL TRANSIENT	X	X
	— PRESSURIZATION TRANSIENT	X	X
	● NOZZLE-TO-CASE JOINT		
	— FLOW AND THERMAL TRANSIENT	X	X
	— PRESSURIZATION TRANSIENT	X	X
	● MNASA MOTOR		
AFE	— WITH BLAST TUBE	X	
	— WITHOUT BLAST TUBE	X	
	● AEROTHERMAL ENVIRONMENTS		
	— DSMC	X	
	— NS	X	X
SPACE STATION	● CONTAMINATION TRACKING	X	
	● ECLSS MODULE AND NODE	X	X
ADVANCED PROGRAM DEVELOPMENT	● EXTERNAL TANK GAMMA RAY IMAGING TELESCOPE	X	

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

VISCOUS ANALYSIS OF THE ATD OXYGEN AND FUEL PUMP DISCHARGE VOLUTES

- **OBJECTIVE: TO OBTAIN SIDELOADS DUE TO HIGH PRESSURE PUMP VOLUTES.**
- **JUSTIFICATION: ACCURATE SIDELOAD PREDICTION IS CRITICAL FOR SUCCESSFUL BEARING DESIGN. WATER-FLOW RIG DATA AVAILABLE FOR OXYGEN PUMP VOLUTE. NO DATA AVAILABLE FOR FUEL PUMP VOLUTE UNTIL 3RD HOTFIRE TEST SERIES OF TURBOPUMP.**
- **APPROACH: USED FDNS2D TO PERFORM INCOMPRESSIBLE, 2-D TURBULENT ANALYSIS OF VOLUTE GEOMETRY. JACOBIAN MODIFIED TO ACCOUNT FOR 3-D CROSS-SECTION AREA. USING 511 X 97 GRID POINTS FOR 5-VANED, DUAL EXIT LOX VOLUTE. USING 481 X 79 GRID FOR 13-VANED SINGLE EXIT FUEL VOLUTE.**
- **RESULTS/IMPACT: 2-D RESULTS: LOX VOLUTE ANALYSES MATCHED RESULTANT LOAD (700 LBS.) BUT NOT DIRECTION. FUEL VOLUTE RESULTS INDICATE VERY SMALL LOADS (< 100 LBS.). PSEUDO 3-D ANALYSIS STILL UNDERWAY.**

LOX VOLUTE RESULTS
PRESSURE CONTOURS (psi)

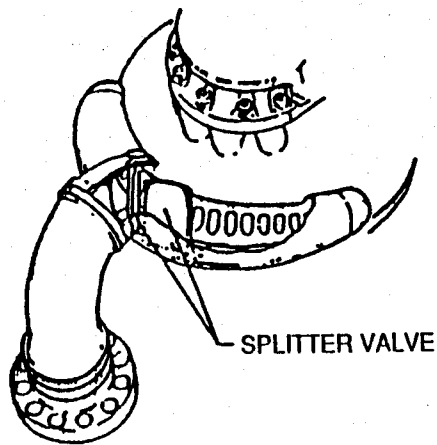


CONTOUR LEVELS

2500.000
2530.000
2560.000
2590.000
2620.000
2650.000
2680.000
2710.000
2740.000
2770.000
2800.000
2830.000
2860.000
2890.000
2920.000
2950.000
2980.000
3010.000
3040.000
3070.000
3100.000
3130.000
3160.000
3190.000
3220.000
3250.000
3280.000
3310.000
3340.000
3370.000
3400.000
3430.000
3460.000
3490.000
3520.000
3550.000
3580.000
3610.000
3640.000
3670.000
3700.000
3730.000
3760.000
3790.000

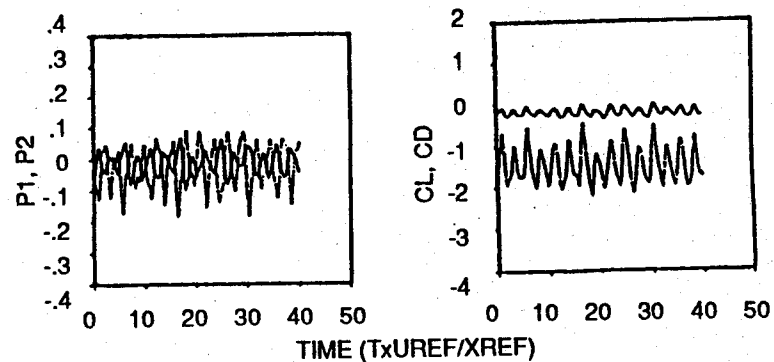
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

MAIN INJECTOR LOX INLET CFD RESULTS

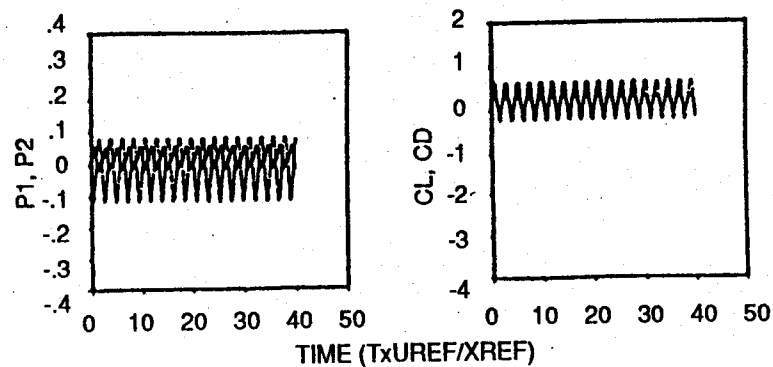


A. MAIN INJECTION LOX INLET

FLIGHT CONFIGURATION $Re = 10^6$; $S_t = .375$ (3800 Hz)



FIX (BEVELED TRAINING EDGE) $Re = 10^6$; $S_t = .445$ (4600 Hz)



B. PRESSURE/LOAD VARIATION WITH TIME

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

NOZZLE/MCC MISMATCH FLOW FIELD ANALYSIS

OBJECTIVES

- DETERMINE TEMPERATURE IN G15 CAVITY FOR VARIOUS MISMATCH CONFIGURATIONS
- ASSESS SENSITIVITY OF CAVITY TEMPERATURE TO VARIATIONS IN MAXIMUM PROTRUSION, CIRCUMFERENTIAL DIFFERENTIALS IN PROTRUSION, AND GAP
- DEVELOP ACCEPT/REJECT CRITERIA FOR FUTURE ENGINE CONFIGURATIONS

APPROACH

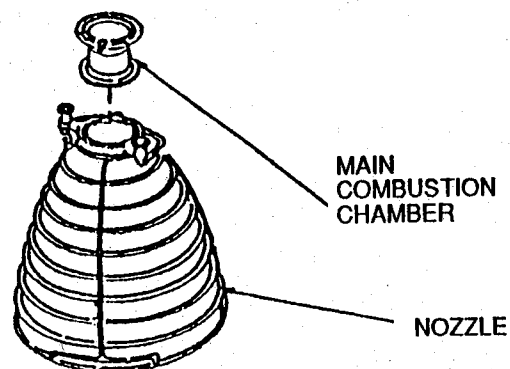
- CFD SIMULATIONS OF 2D, CLOSED CAVITY FLOW (APPROXIMATE GEOMETRY)
 - ASSESS CAVITY PRESSURE AS A FUNCTION OF PROTRUSION
 - DEDUCE APPROXIMATE CIRCUMFERENTIAL FLOW AS A FUNCTION OF MAXIMUM PROTRUSION
- CFD SIMULATIONS OF 2D CAVITY WITH MASS FLOW OUT OF CAVITY BASE (EXACT GEOMETRY)
 - PERFORM PARAMETER SENSITIVITY STUDY
 - DETERMINE FOR WHAT PARAMETER VALUES THE CAVITY TEMPERATURE REACHES THE CRITICAL TEMPERATURE AT WHICH HEAT CONDUCTION AT NOZZLE LIP WILL CAUSE BLUEING OF SEAL

PROGRESS IMPACT

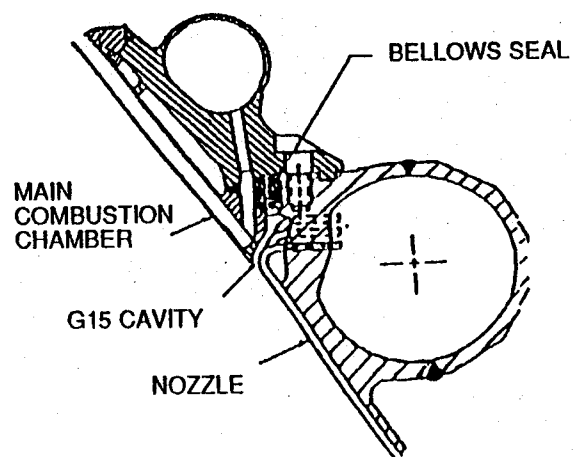
- RESULTS COMMUNICATED TO CHIEF ENGINEER FOR SSME AND TO HARDWARE CONTRACTOR
- RESULTS USED IN DEVELOPMENT OF ACCEPT/REJECT CRITERIA

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

NOZZLE/MCC MISMATCH CFD RESULTS

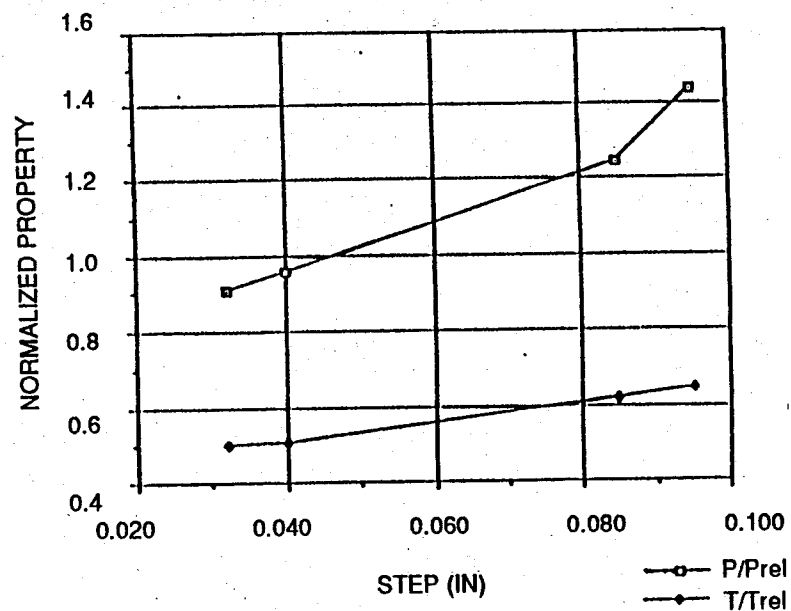


A. ENGINE CONFIGURATION



B. NOZZLE/MCC MISMATCH JOINT

STEP PARAMETRIC



C. PRESSURE/TEMPERATURE VARIATION WITH STEP HEIGHT

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

CURRENT PROGRAMS

Development Programs	KEY PROGRAM MILESTONES												
	CY 87	CY 88	CY 89	CY 90	CY 91	CY 92	CY 93	CY 94	CY 95	CY 96	CY 97	CY 98	CY 99
Advanced Launch System (STEP)	▽ Start • A		▽ Start • B		▽ Start • C/D				▽ Start CDR			▽ 1st Flight	
	CFD ACTIVITIES												
Advanced Solid Rocket Motor				▽ IPDR	▽ PDR		▽ ICDR	▽ CDR					
							●	●	●	●	●	●	
	CFD ACTIVITIES												
Aeroassist Flight Experiment		▽ PRR	▽ PDR	▽ CDR				▽ FLIGHT					
	CFD ACTIVITIES												
Space Station Freedom ECLSS		▽ PRR		▽ PDR		▽ CDR			▽ PMC				
	CFD ACTIVITIES												
Alternate Turbopump Development	▽ PRR	▽ PDR			▽ CDR			▽ DCR					
	CFD ACTIVITIES												

VALIDATION ACTIVITIES

- **BENCHMARK VALIDATION PLAN**

- **LEVEL I: FUNDAMENTAL FLOWS**
- **LEVEL II: SUBCOMPONENTS**
- **LEVEL III: INTERACTIVE COMPONENTS/SYSTEMS**
- **END PRODUCTS**

- **CONSORTIUM**

STATUS OF MSFC CFD APPLICATION VALIDATION ACTIVITIES

● LEVEL I: FUNDAMENTAL FLOWS

CODE	FLAT PLATE	CHANNEL FLOW	STRAIGHT PIPE	CIRCULAR PIPE BEND	RECT PIPE BEND	BACKWARD FACING STEP
INS3D	-	1	1	1	2	1
INS3DLU	1	2	1	2	1	2
INS3DUP	1	1	2	2	2	2
CMINT	-	2	1	2	1	2
FDNS	3	3	4	4	2	3
REFLEQS	1	3	1	2	-	2
MAST	-	-	-	1	-	1

* NUMBER OF PEOPLE WORKING PROBLEM

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

BENCHMARK VALIDATION PLAN

● LEVEL 2 – SUBCOMPONENTS

CASCADE
DISK CAVITY
SINGLE INJECTOR
UNSTEADY FLOW OVER CYLINDER
AXISYMMETRIC DIFFUSER
CURVED DUCT WITH TURNING VANES
HIGH CURVATURE DUCT WITH SEPERATION
ROTATING DUCT
ROTATING CURVED DUCT
NOZZLE
CYLINDER WITH BLOWING WALLS
SLOTTED CYLINDER WITH BLOWING WALLS

● LEVEL 3 – INTERACTIVE COMPONENTS OR SYSTEMS

HOT GAS MANIFOLD PILOT MODEL
UNSTEADY TURBINE STAGE
MAIN INJECTOR/NOZZLE/PLUME
UNSTEADY PUMP STAGE
INLET AND DISCHARGE VOLUTES
AEROASSIST FLIGHT EXPERIMENT
BEARING CAVITY MODEL

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

BENCHMARK VALIDATION PLAN

● DOCUMENTATION REQUIREMENTS

- OBJECTIVE: DESCRIBE PROBLEM AND THE FOCUS/EXPECTATION OF THE CFD COMPUTATION
- COMPUTATIONAL STRATEGY: PROBLEM DEFINITION, PROBLEM GOAL, BOUNDARY CONDITIONS, INITIALIZATION, USER ISSUES
- DATA BASE: DEFINITION OF GEOMETRY, Re , M , DATA SOURCE, ETC.
- COMPUTER REQUIREMENTS: CPU TIME, STORAGE REQUIREMENTS, SSD REQUIREMENTS
- LESSONS LEARNED: DESCRIPTION OF FALSE STARTS, GRID SENSITIVITY, INITIALIZATION, B.C. PROBLEMS.
- SUMMARY: STRENGTH/WEAKNESS IN COMPUTATION VS. DATA COMPARISON. EVALUATION OF CODE FOR POTENTIAL HARDWARE APPLICATION.

DOCUMENTATION SHOULD BE CONCISE.

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

BENCHMARK VALIDATION PLAN

● END PRODUCTS:

- DIMENSIONLESS EVALUATION CRITERIA
- ERROR ESTIMATION METHODOLOGY
- ASSESSMENT OF CODES BASED UPON CRITERIA AND ERROR ESTIMATION

● FOCUS OF CRITERIA:

- FLOW RANGES (MACH NUMBER, REYNOLDS NUMBER --)
- CODE (SOLUTION ALGORITHM, CODE ARCHITECTURE, BOUNDARY CONDITIONS --)
- FLOW PROCESS MODELS (TURBULENCE MODEL, MULTISPECIES --)
- USER SPECIFIC EFFECTS (GRID GENERATING, INITIALIZATION --)

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

CONSORTIUM

● CONSORTIUM OBJECTIVES

● FOCUS CFD APPLICATIONS IN PROPULSION

● TECHNOLOGY ACQUISITION PHASE

- DIRECT BASELINE PROGRAM TOWARDS IMPROVED ACCURACY, STABILITY, AND EFFICIENCY**

● LARGE SCALE SUBSYSTEM TECHNOLOGY VALIDATION

- STIMULATE CFD VALIDATION TOWARDS PROPULSION FLOWS**
- DIRECT APPLICATIONS CODES TOWARD DESIGN TOOLS AND ADVANCED TECHNOLOGY HARDWARE CONCEPTS**

● IDENTIFY NATIONAL CFD PROPULSION REQUIREMENTS

● STIMULATE A FORUM FOR GOVERNMENT, INDUSTRY, AND UNIVERSITY INTERACTIONS

● ENCOURAGE INDUSTRY TO PARTICIPATE IN CFD DEVELOPMENT WITH IRAD FUNDS

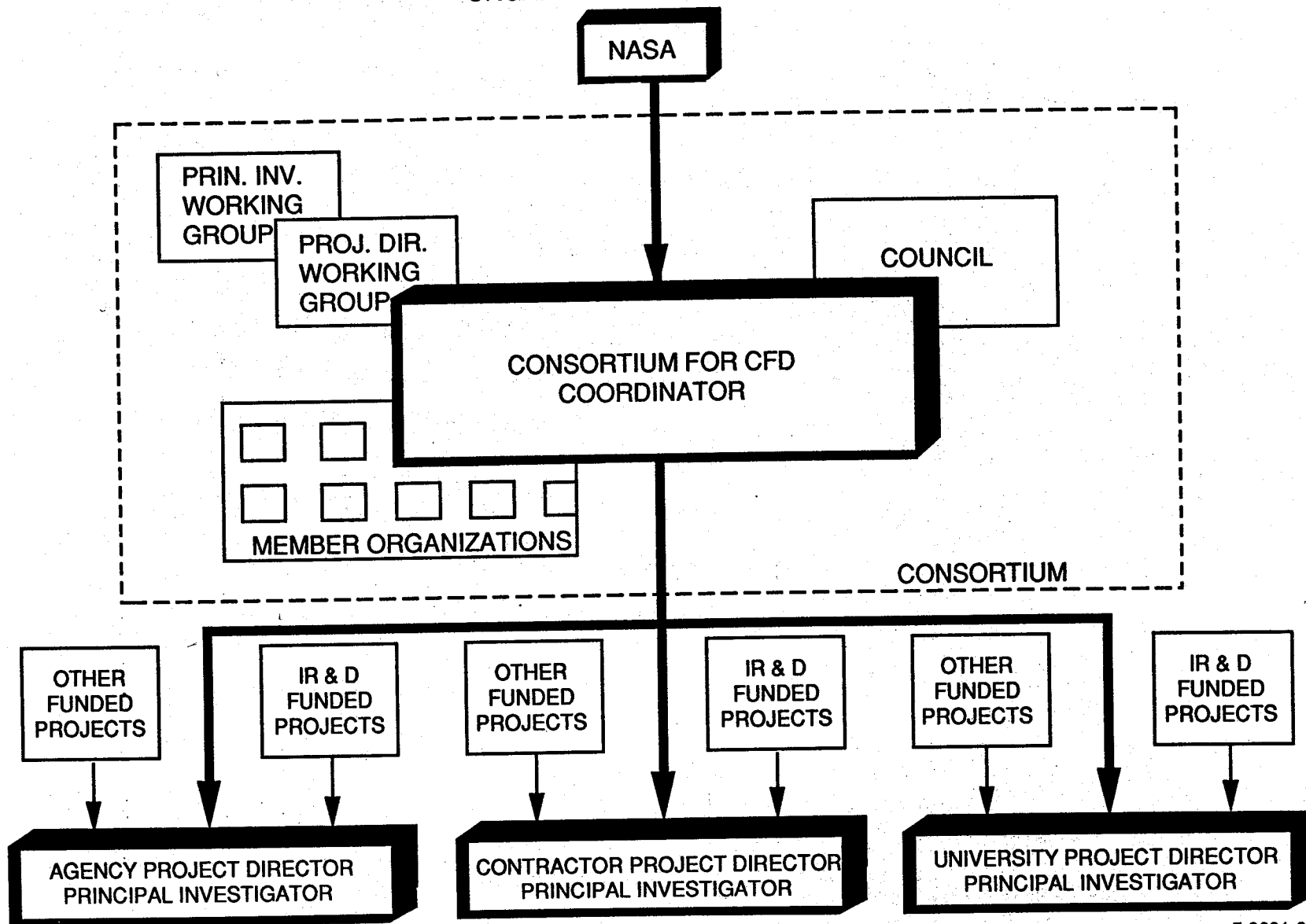
● PROVIDE SYNERGISM IN THE CFD COMMUNITY

● PROVIDE PEER REVIEW OF CFD PROGRAMS

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

● CONSORTIUM FOR CFD APPLICATIONS IN PROPULSION TECHNOLOGY

ORGANIZATIONAL STRUCTURE



STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

ADVANCED TECHNOLOGY HARDWARE

● CONSORTIUM TEAMS

FOCUS DEVELOPMENT OF CFD METHODOLOGY
AND
DEVELOP ADVANCED TECHNOLOGY HARDWARE CONCEPTS

TURBOMACHINERY

- TURBINES
- PUMPS

COMPLEX FLOW PATHS

- DUCTS
- COOLANT FLOWS
- ROTATING CAVITIES

CASTINGS

COMBUSTION DRIVEN FLOWS

SOLID ROCKET MOTORS

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

● CONSORTIUM TEAM VALIDATION ACTIVITIES

- TURBINE TEAM

- CASCADES**
- ROTOR STATOR**
- SSME**

- PUMP TEAM

- ROCKETDYNE INDUCER**

- COMPLEX FLOW PATHS

- DISK CAVITIES**
- HGM PILOT MODEL**
- DUCT FLOWS**

- SRM TEAM

- CSD CYLINDER**
- SLOTTED CYLINDER**
- SRM PILOT MODEL**

- COMBUSTION TEAM

- H₂ MIXING**
- DUMP COMBUSTION**
- NOZZLE FLOW**

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

CONSORTIUM

● TURBINE STAGE TECHNOLOGY TEAM

OBJECTIVE: DEVELOPMENT, ENHANCEMENT, VALIDATION, AND DEMONSTRATION OF CFD TOOLS FOR TURBINE STAGE DESIGN

- APPROACH:
- ENHANCE EXISTING DATA BASE (COLD FLOW RIG AND HOT-FIRE SYSTEM TESTS), DEVELOP/ENHANCE CFD MODELS AND CODES, EXPERIMENTALLY DEMONSTRATE CFD-PREDICTED DESIGN IMPROVEMENTS
 - ASSEMBLE TURBINE TECHNOLOGY EXPERTS REPRESENTING DIFFERENT AREAS OF EXPERTISE. POOL TECHNICAL RESOURCES AND EXPERIENCE BASES TO BRING VARIOUS PERSPECTIVES AND ABILITIES TO BEAR ON TURBINE STAGE ISSUES AND ON ACCOMPLISHMENT OF TEAM OBJECTIVE
 - DEVELOP AND IMPLEMENT PLAN WHICH COORDINATES MSFC-SUPPORTED TURBINE STAGE ANALYSIS AND TEST ACTIVITIES
 - IDENTIFY MILESTONES, DELIVERABLE PRODUCTS, AND TARGET DATES
 - IDENTIFY ADDITIONAL ACTIVITIES REQUIRED TO ACCOMPLISH TEAM OBJECTIVE
 - INTERACT THROUGH REGULAR MEETINGS TO STATUS, CROSS-FERTILIZE, CRITIQUE, AND/OR DIRECT EACH ACTIVITY

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

CONSORTIUM

● TURBINE STAGE TECHNOLOGY TEAM

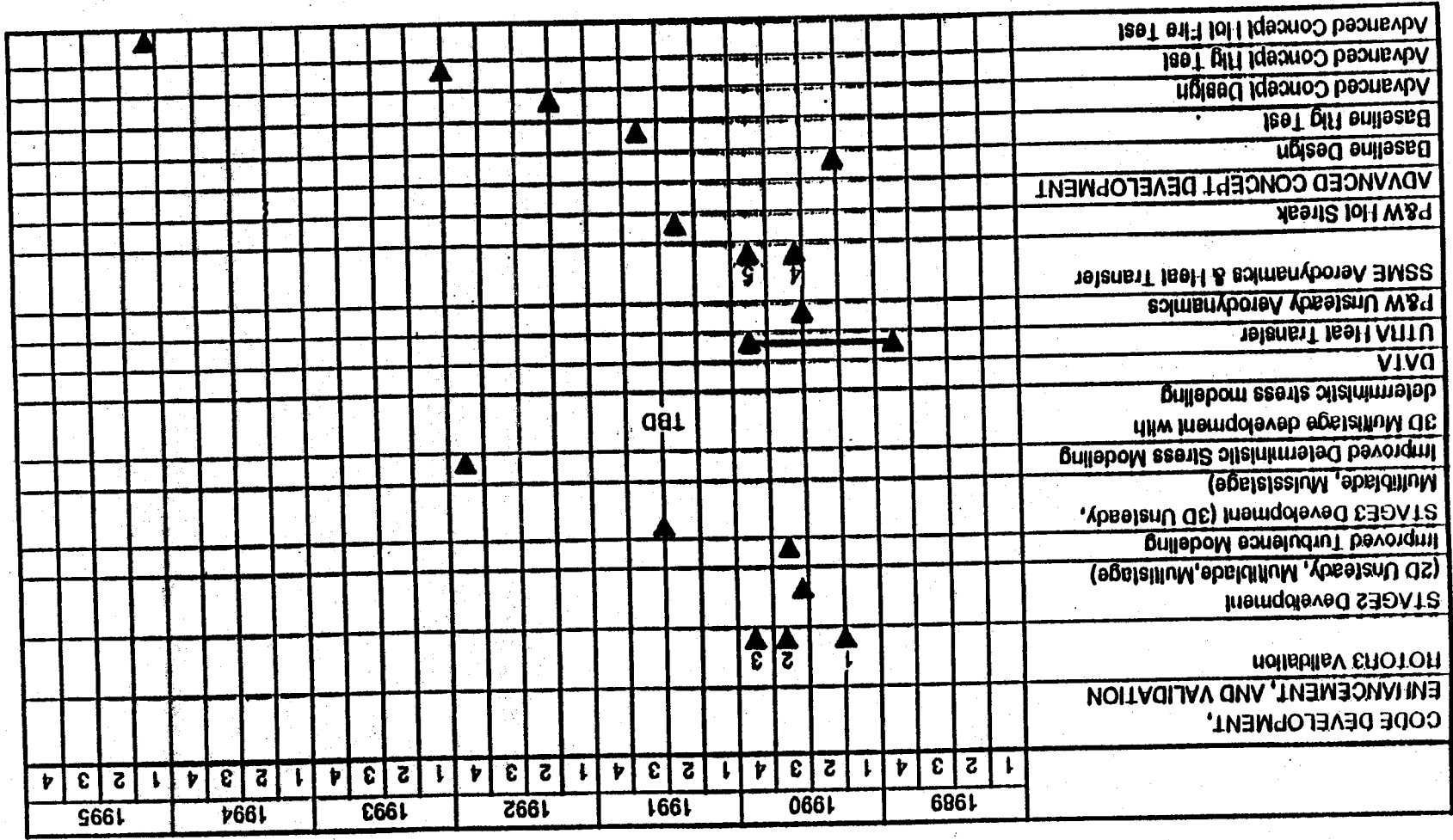
STATUS: 5 MEETINGS HELD TO DATE—4/28/89, 8/23/89, 11/1/89, 1/17/90, 4/19/90

MEMBERSHIP:

- NASA MARSHALL SPACE FLIGHT CENTER (MSFC)
- NASA AMES RESEARCH CENTER (ARC)
- NASA LEWIS RESEARCH CENTER (LARC)
- ROCKETDYNE (RDYN)
- PRATT & WHITNEY (P&W)
- AEROMET
- GENERAL ELECTRIC COMPANY
- UNITED TECHNOLOGIES RESEARCH CENTER (UTC)
- CALSPAN—UNIVERSITY OF BUFFALO RESEARCH CENTER (CUBRC)
- ROTODATA, INC.
- SCIENTIFIC RESEARCH ASSOCIATES (SRA)
- UNIVERSITY OF ALABAMA—HUNTSVILLE (UAH)
- PENNSYLVANIA STATE UNIVERSITY (PSU)
- MISSISSIPPI STATE UNIVERSITY (MSU)

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

CONSORTIUM - TURBINE TEAM KEY MILESTONES

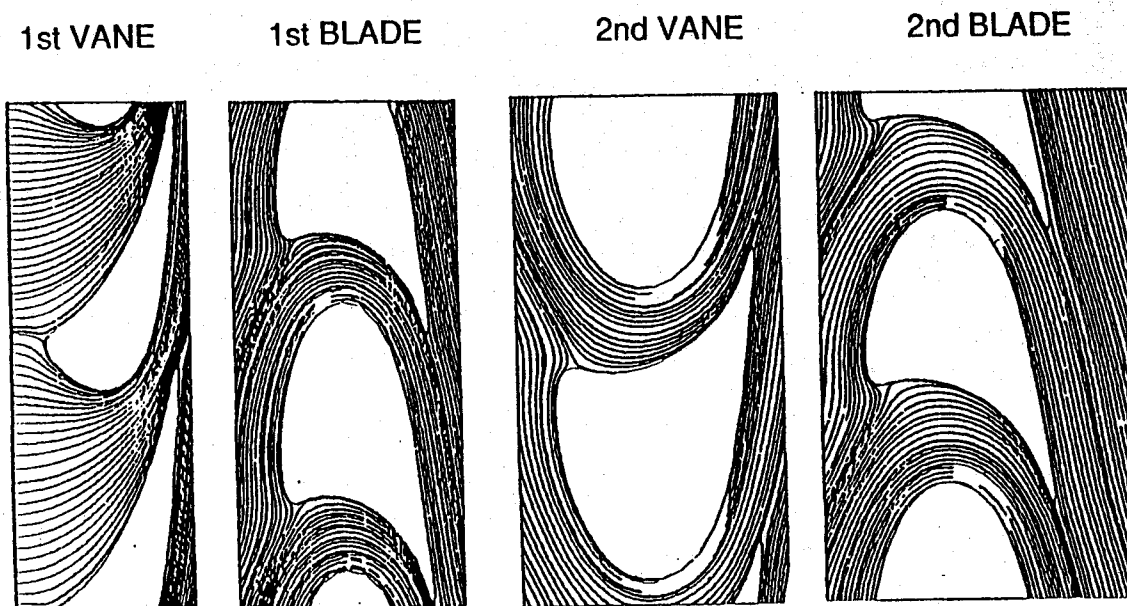


KEY: 1. Validation against steady aerodynamic and heat transfer data
 2. Validation against unsteady aerodynamic data
 3. Validation against SSME or advanced concept data
 4. Steady and unsteady aerodynamic data, unsteady heat transfer data
 5. ATD steady aerodynamic data

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

CONSORTIUM - TURBINE TEAM

- BASELINE TURBINE MIDSPAN CONTOURS/CALCULATED STREAMLINES



NOTE: ROW TO ROW SCALES ARE DIFFERENT

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

CFD REQUIREMENTS

- GRID GENERATION
 - ELECTRONIC SURFACE GENERATORS COMPATIBLE WITH CFD CODES
 - INTERACTIVE GRID PACKAGES WITH INTERROGATION SCHEMES
 - ADAPTIVE GRID PACKAGES
- EVALUATION CRITERIA AND ERROR ANALYSIS METHODOLOGY
 - STANDARDIZED DIMENSIONLESS PARAMETERS
- VALIDATION EXPERIMENTS
- CODES
 - BENCHMARKING USING STANDARDIZED METHODS
 - MULTIBLOCKING OR ZONAL STRATEGY
- FLOW PROCESS MODELS
 - TURBULENCE AND TRANSITION
 - MULTISPECIES
 - CHEMISTRY WITH AND WITHOUT REACTIONS
 - MULTIPHASE
 - ATOMIZATION
 - EVAPORATION
 - MIXING
- POST PROCESSING
 - DEMONSTRATION OF CONSERVATION PROPERTIES
 - ERROR ANALYSIS
- GRAPHICS
 - INTERACTIVE
 - THREE DIMENSIONAL FLOW DESCRIPTIONS
 - ANIMATION

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

CFD EXPECTATIONS

- DIRECT HARDWARE DESIGN UTILIZING CFD
 - PROVIDE INITIAL IMPACT IN DESIGN
 - PERFORM DESIGN OPTIMIZATION STUDIES
 - DEVELOP ADVANCED TECHNOLOGY HARDWARE CONCEPTS
- BENCHMARKED/VALIDATED CODES
 - LAMINAR FLOWS
 - TURBULENT FLOWS \bar{u}_i, \bar{p}
 - ACOUSTIC PROBLEMS
 - CERTAIN CLASS OF UNSTEADY PROBLEMS
- GENERAL METHODOLOGY FOR ERROR ESTIMATION AND EVALUATION CRITERIA FOR CODES
- USER FRIENDLY CODES
 - B.S. LEVEL ENGINEER 2-3 YRS EXPERIENCE
 - GUIDELINES FOR CLASSES OF PROBLEMS
 - CAD/CAM/CAE; GEOMETRY GRID GENERATION
 - GENERALIZED BOUNDARY CONDITIONS
 - ALGORITHM/GRID OPTIMIZATION FOR SOLUTION EFFICIENCY
 - ARTIFICIAL INTELLIGENCE/EXPERT SYSTEMS
 - MODULAR CODES
- FLOW ADAPTIVE GRIDS FOR CURRENT CLASS OF PROBLEMS
- MULTIPLE SCALE AND/OR ZONAL TURBULENCE MODELS, MULTIPHASE, MULTISPECIES, COMBUSTION FLOW PROCESS ENGINEERING MODELS EVOLVED FROM EXPERIMENTS AND CFD ANALYSIS

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

SUMMARY

- **CFD BEING APPLIED IN THE HARDWARE DESIGN PROCESS AT MSFC**
 - **FOCUS ON TEAM STRATEGY; RIGHT PROBLEM**
 - **QUICK TURN-AROUND; SENSITIVITY ANALYSIS**
 - **RETROFITTABLE DESIGN OPTIONS**

- **VALIDATION ACTIVITIES PROVIDING COMMUNITY SYNERGISM**
 - **LEVEL I BENCHMARK PLAN IN PLACE / IN PROGRESS**
 - **CONSORTIUM TEAMS INTEGRATE ADVANCED HARDWARE DESIGN AND VALIDATION ACTIVITIES**
 - **FOCUS CFD GOALS AND DEFINE REQUIREMENTS**
 - **INTEGRATE ACTIVITIES, POOL RESOURCES, AND TRANSFER TECHNOLOGY BETWEEN GOVERNMENT, CONTRACTORS, AND UNIVERSITIES**

- **CFD EXPECTATIONS ARE APPLICATIONS-ORIENTED AND REQUIRE SUSTAINED TECHNOLOGY TRANSFER FROM RESEARCH CENTERS**

National Aeronautics and
Space Administration
Lewis Research Center

2nd NASA CFD VALIDATION WORKSHOP



SUMMARY OF LEWIS RESEARCH CENTER VALIDATION EFFORTS

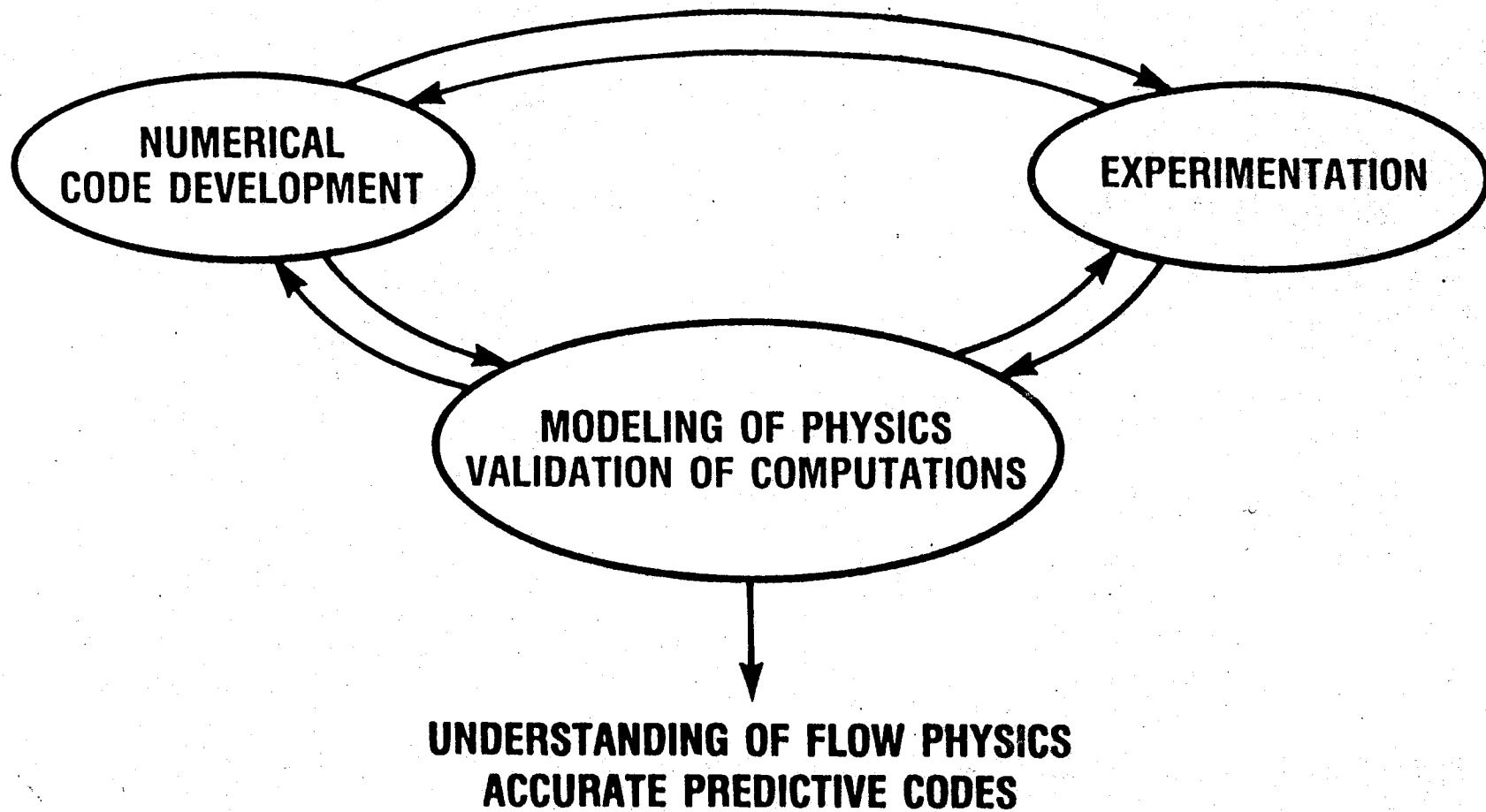
RAYMOND E. GAUGLER
CHIEF, TURBOMACHINERY FLOW PHYSICS BRANCH

62502-36N

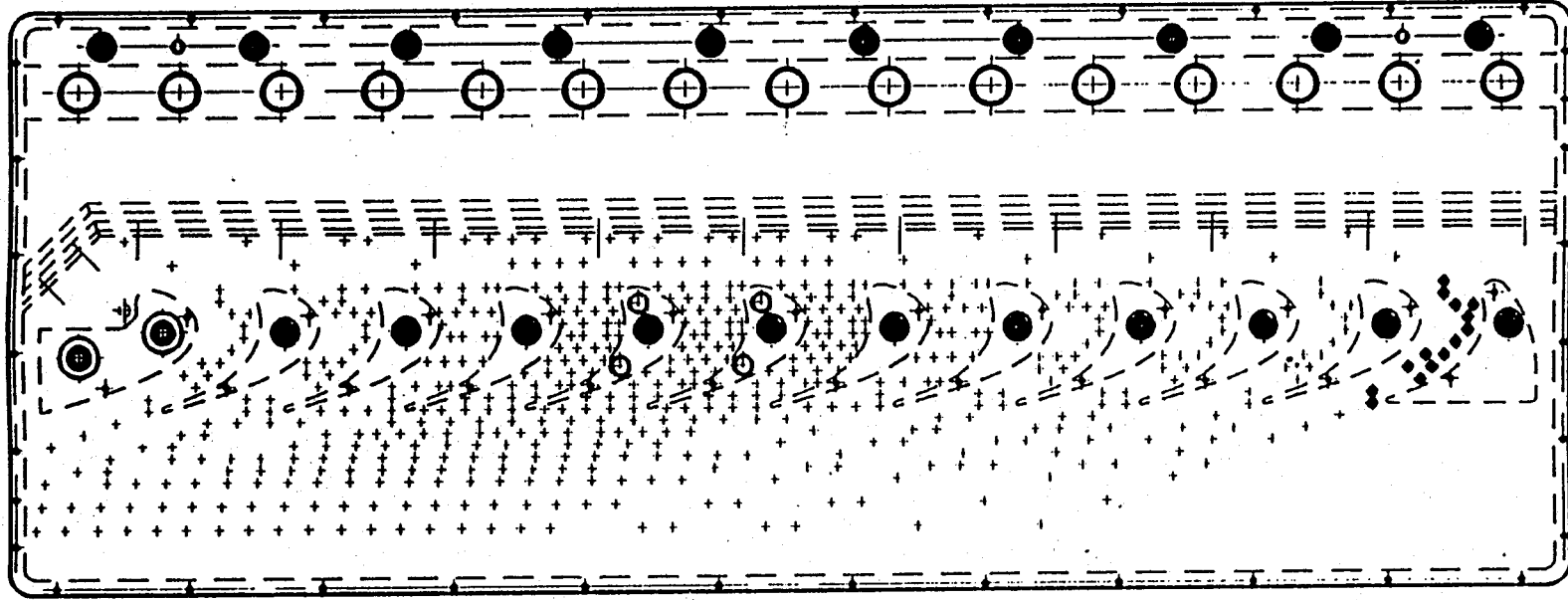
VALIDATION VGF 00

54-34
106674
P-26

CLOSELY COUPLED EXPERIMENTAL AND COMPUTATIONAL RESEARCH

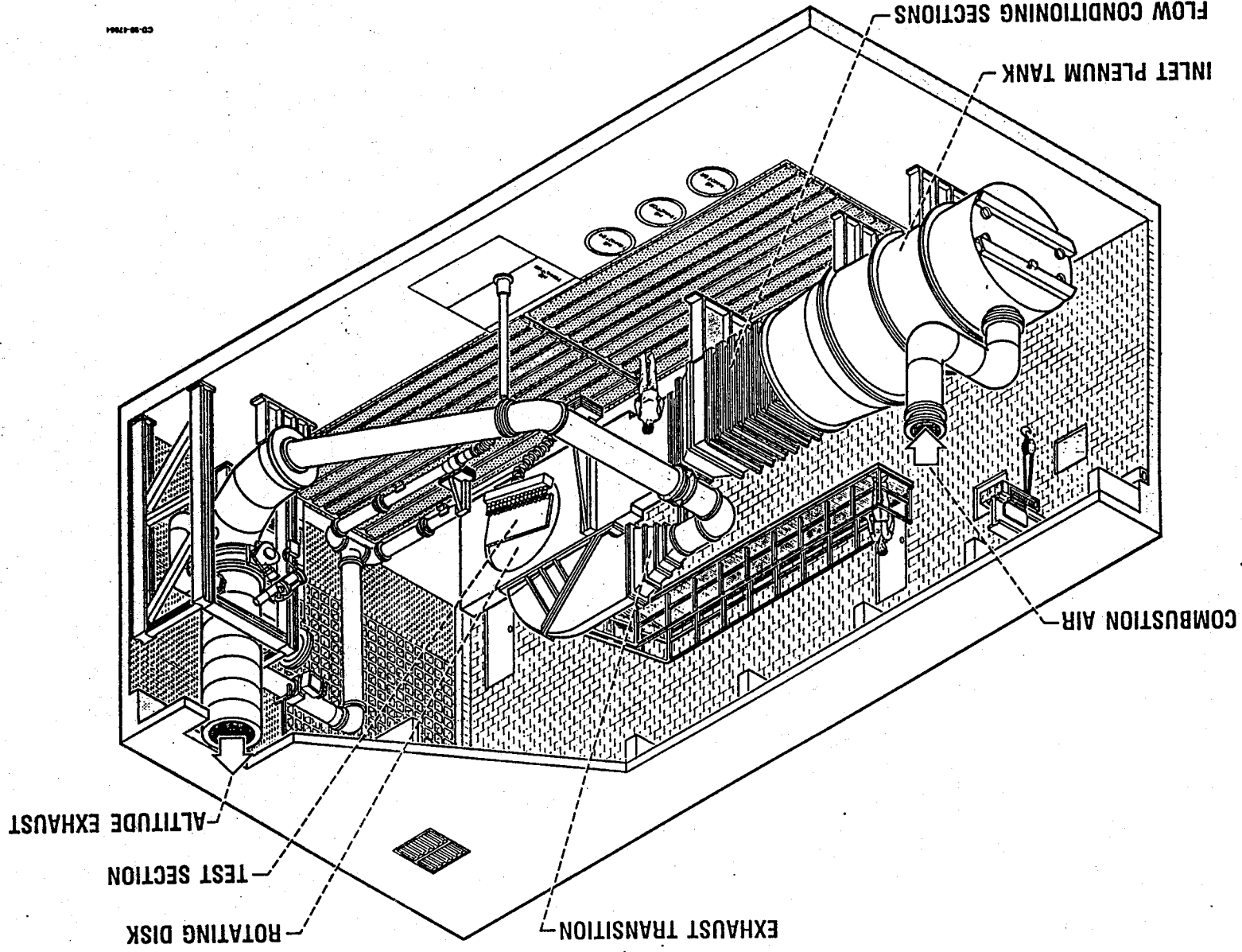


LINEAR TRANSONIC CASCADE



INSTRUMENTATION - PRESSURE TAPS

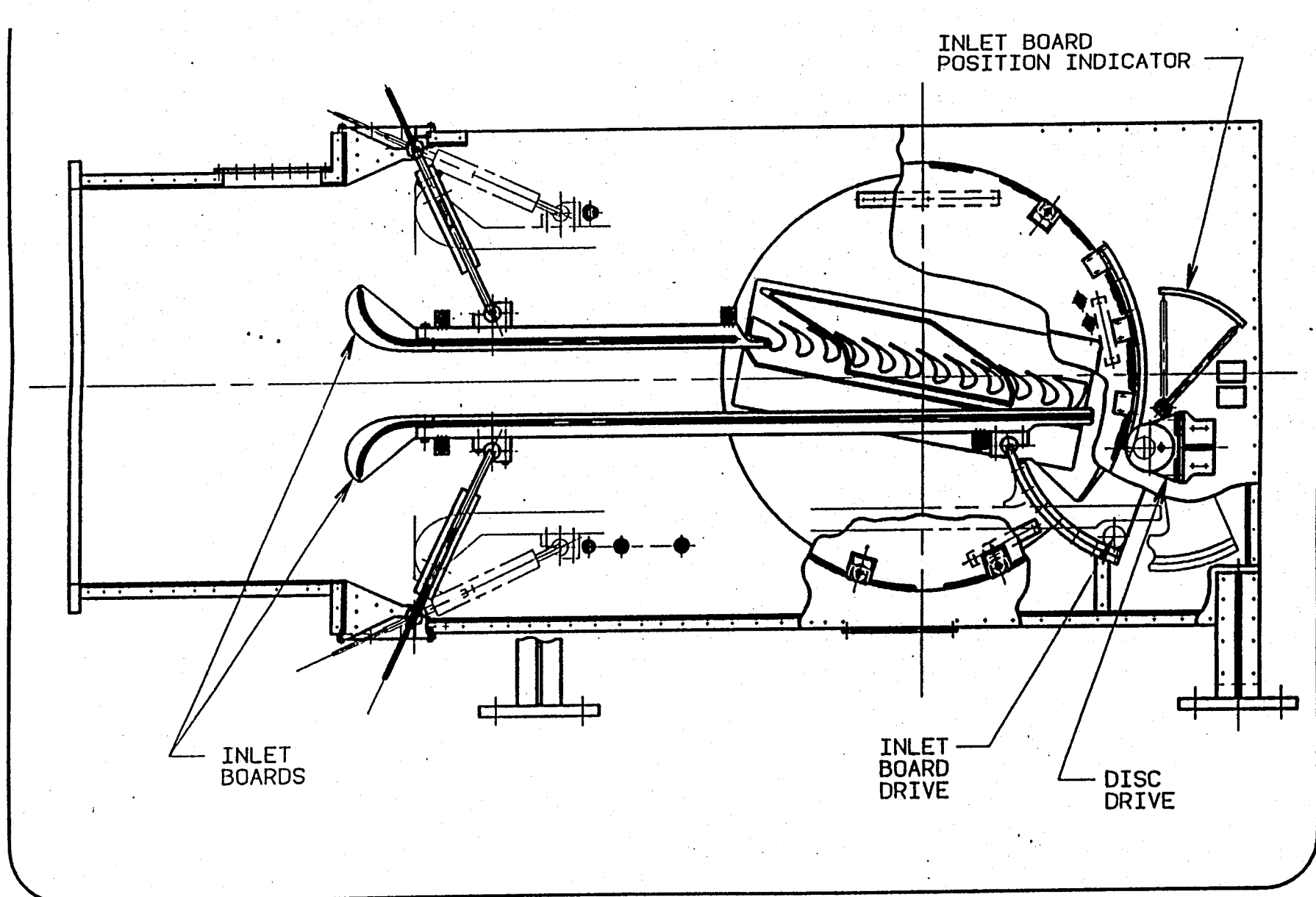
LINEAR TRANSONIC CASCADE TEST CELL CW22 OF THE ENGINE RESEARCH BUILDING

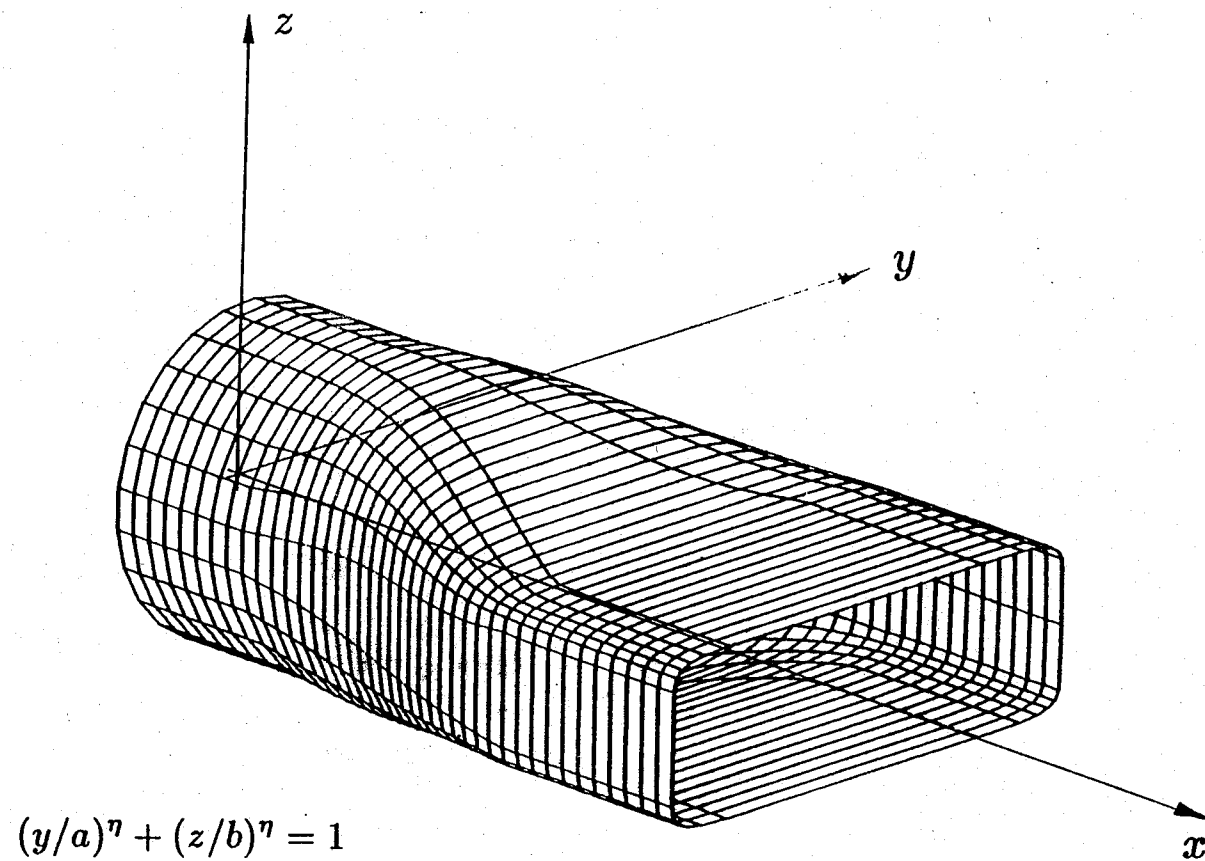


CS-38-1794

INTERNAL FLUID MECHANICS DIVISION
HEAT TRANSFER BRANCH

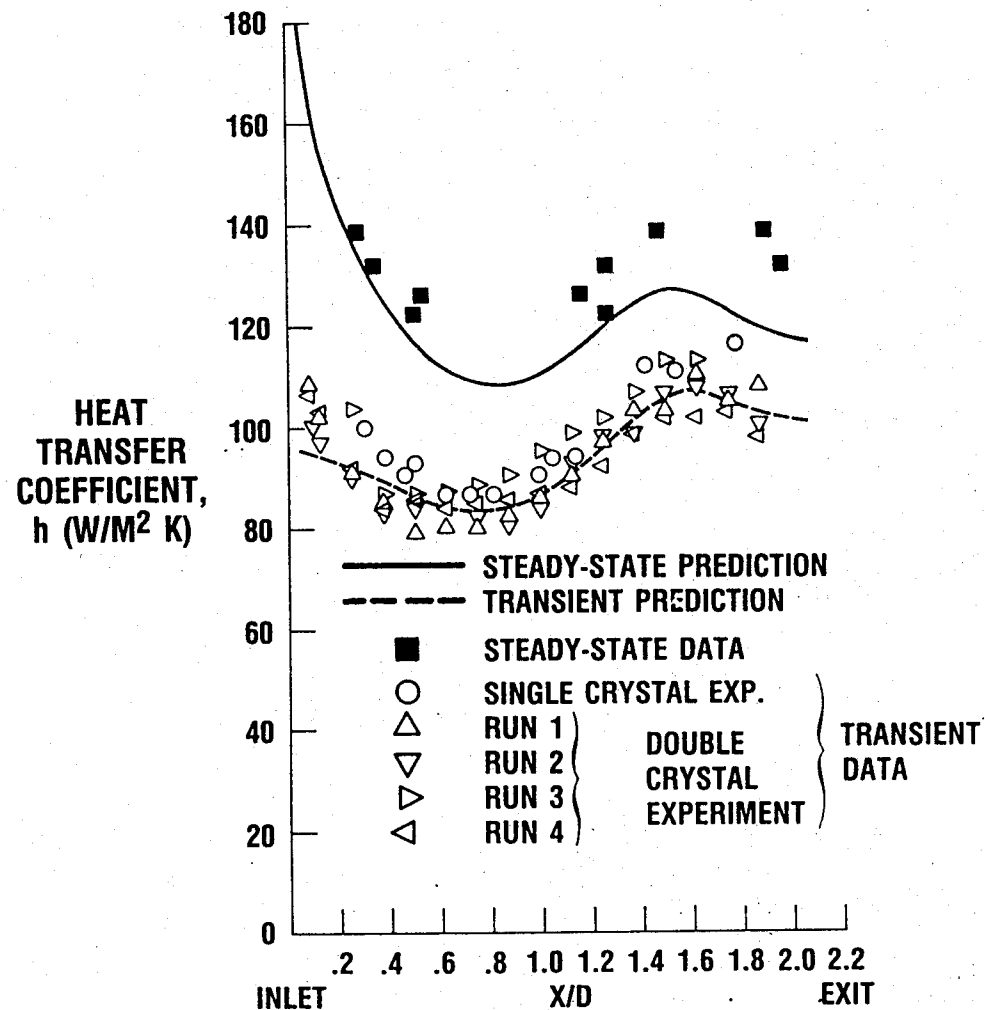
LINEAR TRANSONIC CASCADE





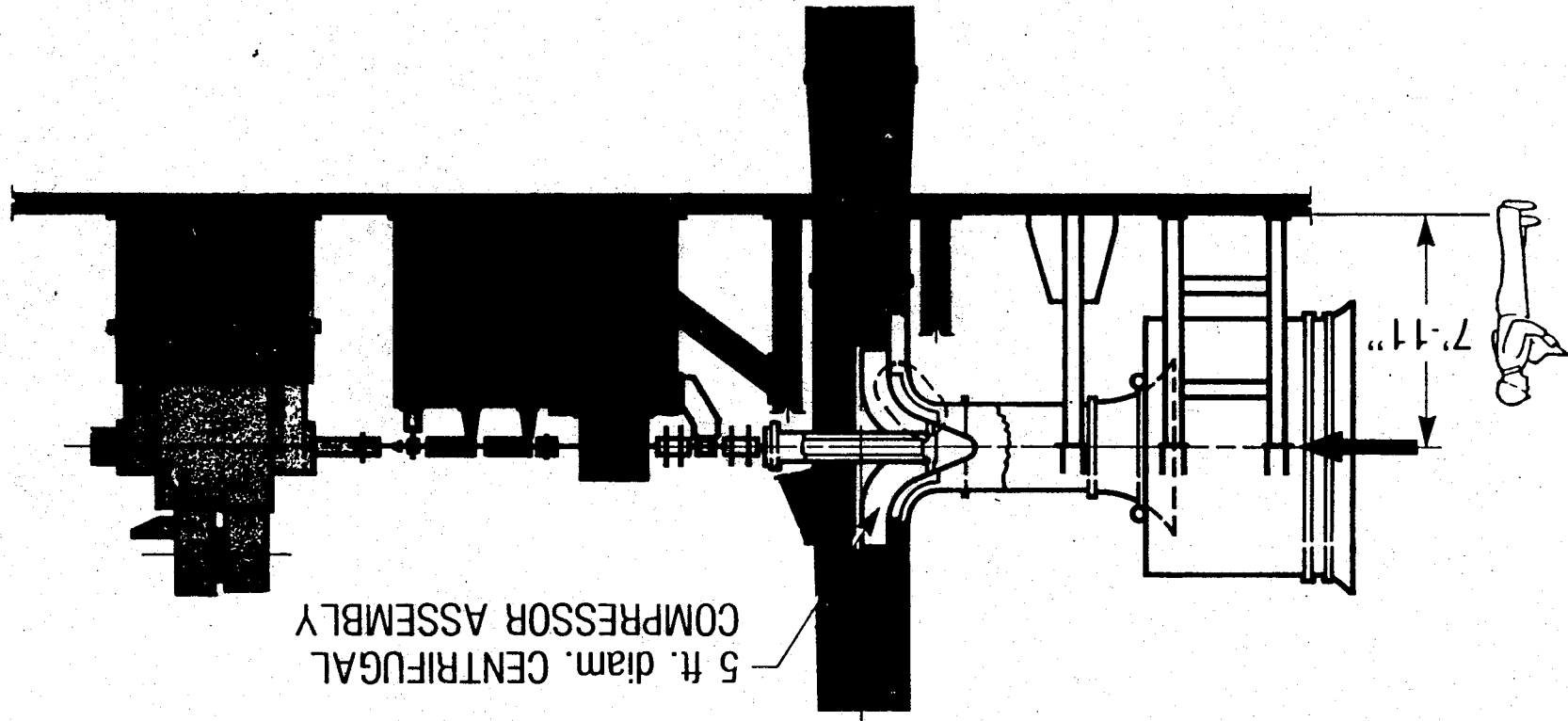
AR410 TRANSITION DUCT

HEAT TRANSFER COEFFICIENTS ALONG DUCT CENTERLINE



EXPERIMENTAL AND PREDICTED HEAT TRANSFER COEFFICIENTS

LOW SPEED CENTRIFUGAL COMPRESSOR BENCHMARK EXPERIMENTS FOR COMPUTATIONAL CODES



CD-84-15176

ORIGINAL PAGE IS
OF POOR QUALITY

82216-88-821

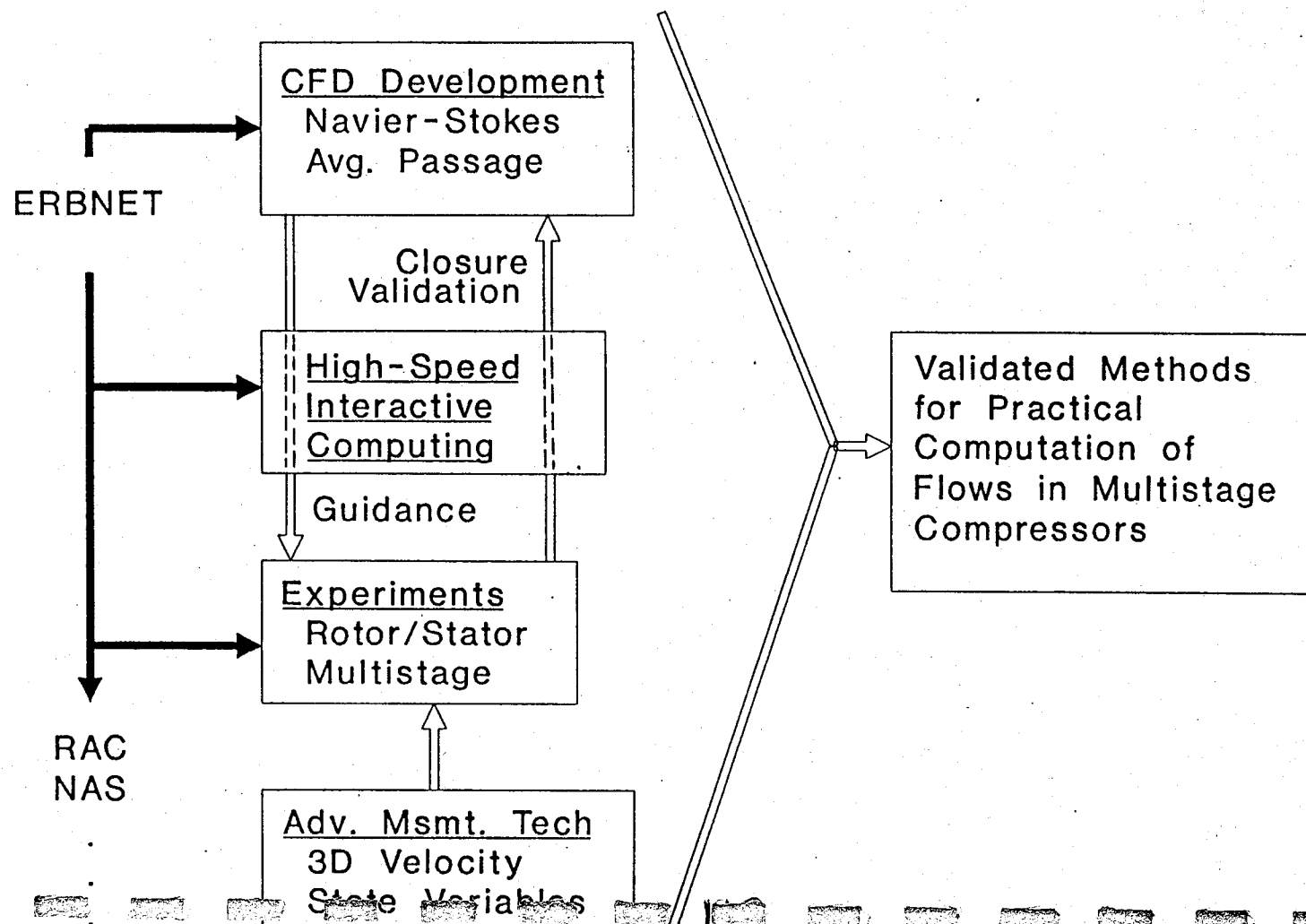
RECEIVED
FBI
JAN 10 1968

C-88-00825

TURBOMACHINERY BLADE ROW INTERACTIONS

MULTI-STAGE COMPRESSOR FLOW PHYSICS

MULTISTAGE COMPRESSOR FLOW PHYSICS PROGRAM



TURBOMACHINERY BLADE ROW INTERACTIONS

MOTIVATION –

Current Designs Based On Time–Averaged
Axisymmetric Analyses

Actual Flow Field Highly Non–Axisymmetric
And Time Dependent

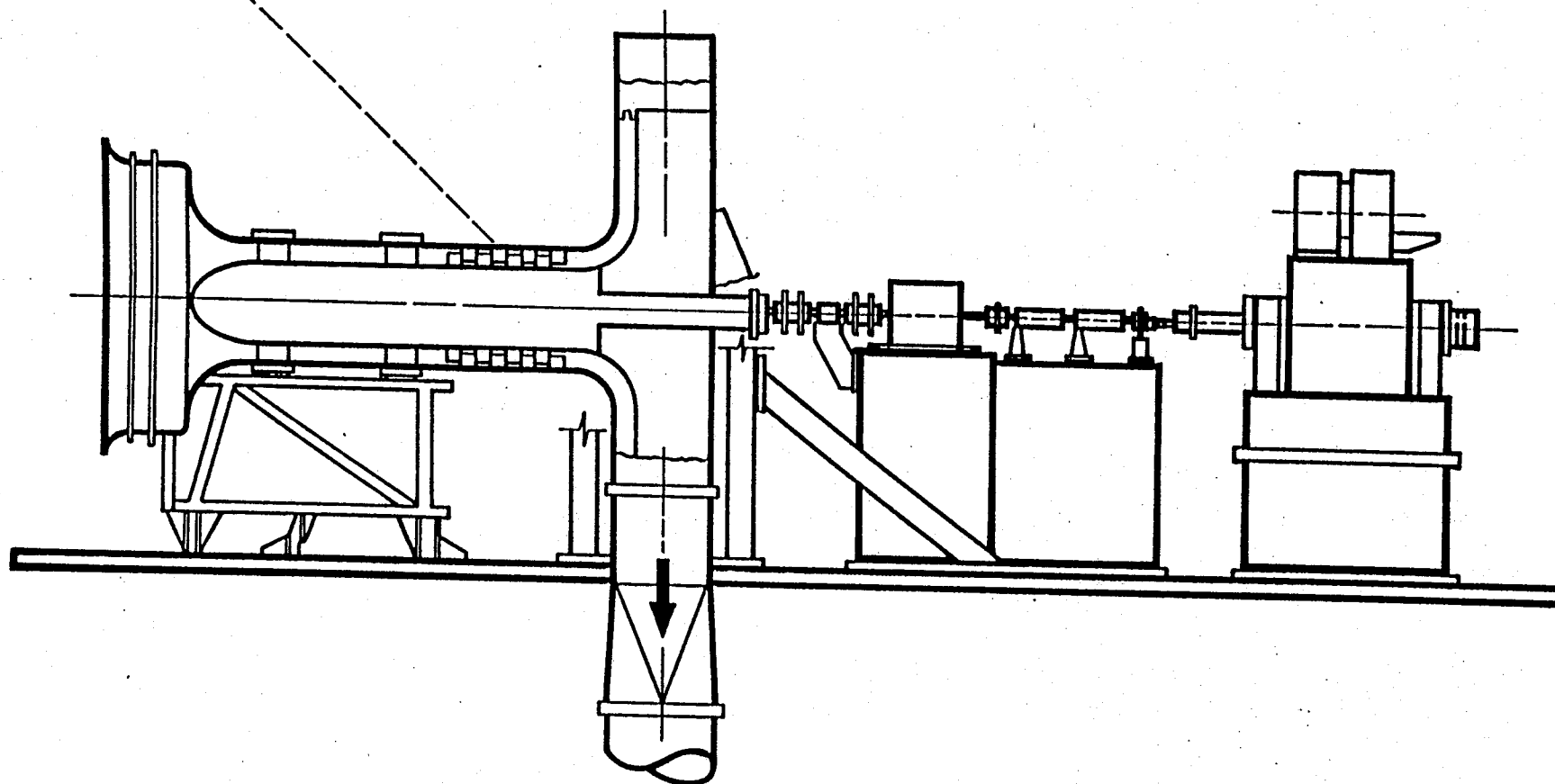
OBJECTIVE

Experimentally Validated Numerical Analysis
Of Flows In Multi–Stage Turbomachinery

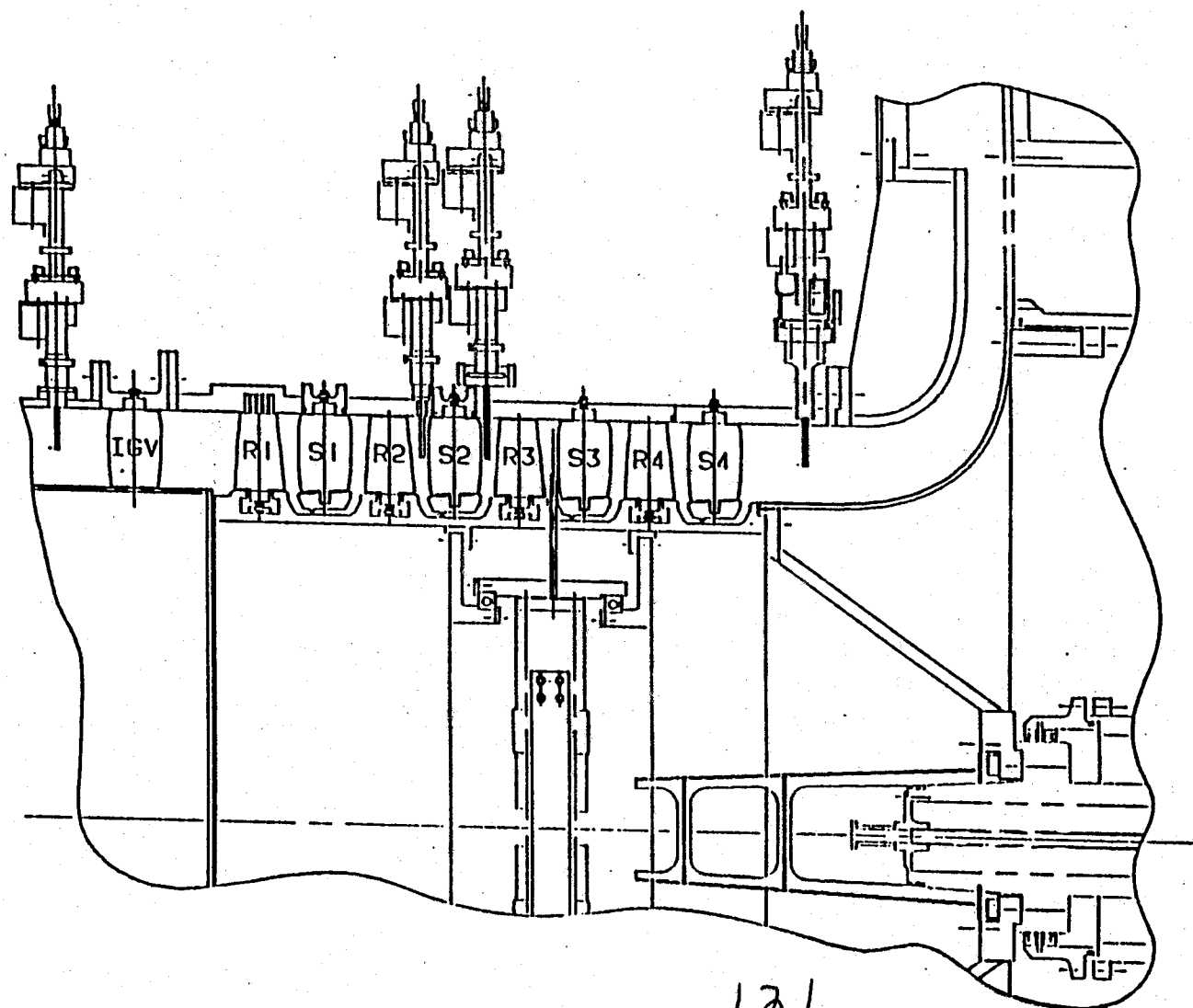
Improved Understanding of Flow Physics
Of Blade–Row Interactions

LOW SPEED MULTISTAGE AXIAL-FLOW COMPRESSOR BENCHMARK EXPERIMENTS FOR COMPUTATIONAL CODES

4 ft. diam. AXIAL-FLOW COMPRESSOR ASSEMBLY



LOW SPEED MULTISTAGE AXIAL COMPRESSOR





MULTISTAGE COMPRESSOR FLOW PHYSICS PROGRAM SUMMARY

- A SYNERGISTIC PROGRAM PULLING TOGETHER CFD, EXPERIMENTS, AND HIGH-SPEED INTERACTIVE COMPUTING TO PROVIDE A NEEDED CAPABILITY
- CFD - UTILIZE AVERAGE-PASSAGE METHOD FOR MULTI-STAGE SIMULATION AND TIME ACCURATE METHODS TO SUPPORT CLOSURE MODELING
- EXPERIMENTS - INITIAL RESULTS FROM HIGH-SPEED, SINGLE STAGES - CLOSURE MODELING AND VALIDATION/ CALIBRATION FROM EXISTING HIGH-SPEED MULTISTAGE AND NEWLY REQUIRED LOW SPEED MULTISTAGE
- MEASUREMENT TECH. - INSTRUMENTATION ADVANCES REQUIRED FOR 3D VELOCITY AND SCALAR MEASUREMENTS IN COMPRESSIBLE FLOWS
- HIGH-SPEED INTERACTIVE COMPUTING - REQUIRED FOR TIMELY CFD AND EXPERIMENTAL INTERACTION

LeRC/IFMD INLET DUCT AND NOZZLE HIGH SPEED VALIDATION EXPERIMENTS

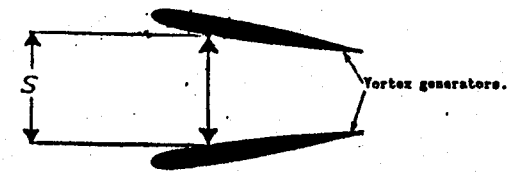
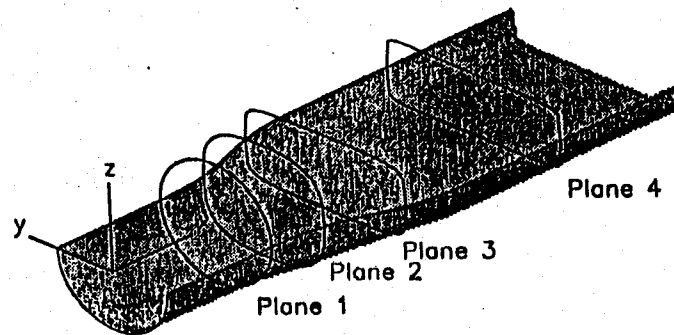
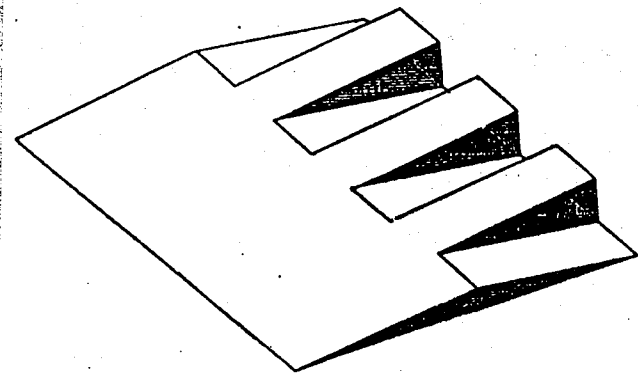
- VALIDATION WORKSHOP EXPERIMENTS
 - CROSSING SHOCKS/BOUNDARY-LAYER INTERACTION
 - LeRC/UNIVERSITY UNSTEADY SHOCK/BOUNDARY-LAYER INTERACTION
- ADDITIONAL VALIDATION EXPERIMENTS
 - HIGH SPEED MIXING
 - TRANSITION DUCTS
 - VORTEX GENERATORS

EXPERIMENTS IN 3-D FLUID MECHANICS

HYPERMIXING

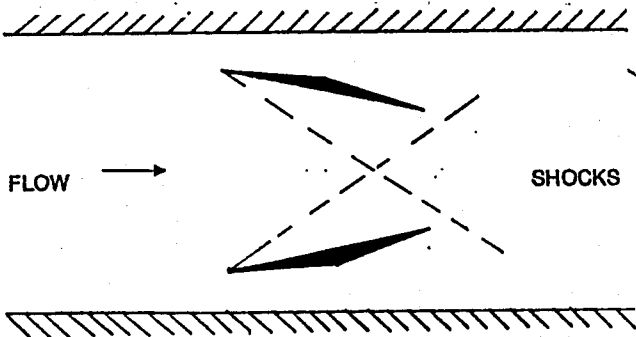
TRAN. DUCT

VORTEX GEN

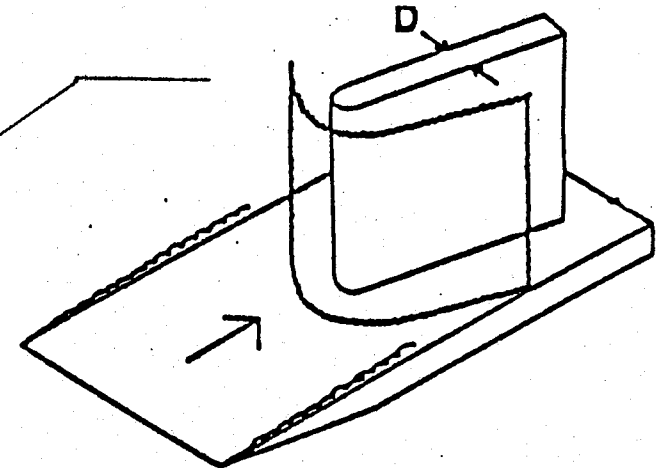


Top view.

CROSSING SHOCK

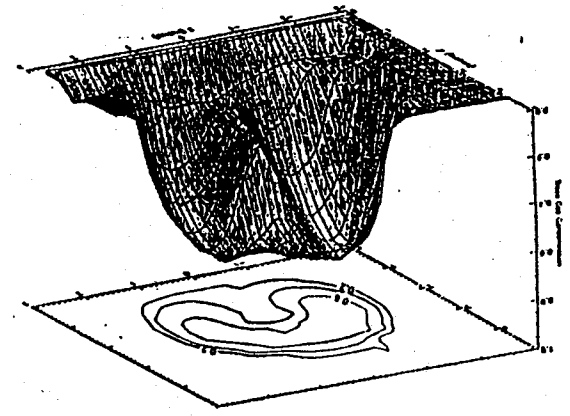


UNSTEADY SHOCK

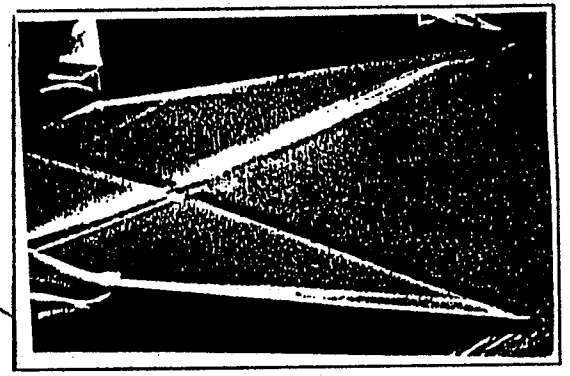


QUANTITATIVE DATA
QUALITATIVE DATA

EXPERIMENTS IN 3-D FLUID MECHANICS

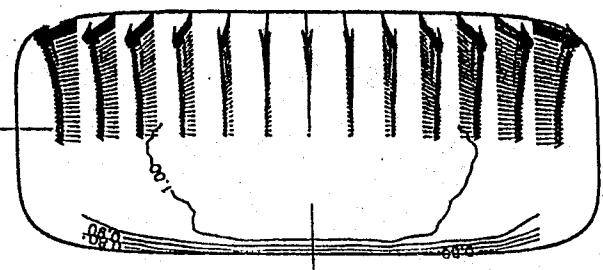


HYPERMIXING

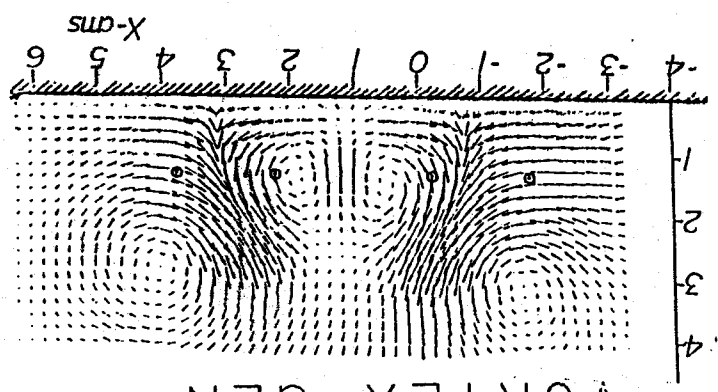


CROSSING SHOCK

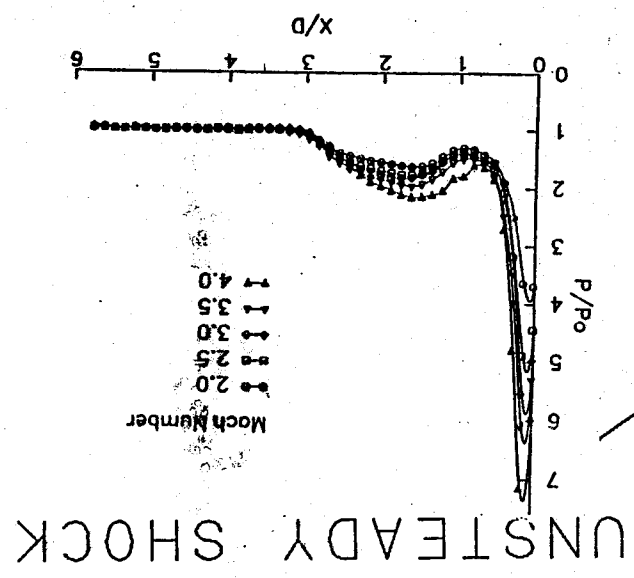
CFD VALIDATION
FLOW PHYSICS



TRAN. DUCT



VORTEX GEN



UNSTEADY SHOCK

125

55-34 P-37
116675

EXPERIMENTAL AND COMPUTATIONAL SURFACE AND FLOW-FIELD RESULTS FOR AN ALL-BODY HYPERSONIC AIRCRAFT

WILLIAM K. LOCKMAN
NASA Ames Research Center
Moffett Field, CA 94035-1000

SCOTT L. LAWRENCE
NASA Ames Research Center
Moffett Field, CA 94035-1000

JOSEPH W. CLEARY
Eloret Institute
Sunnyvale, CA 94087

Second NASA CFD Validation Workshop
NASA Lewis Research Center
July 10-11, 1990

N93-70580

OUTLINE

OBJECTIVE

EXPERIMENT

- **Model**
- **Test Conditions**
- **Measurements**

UPS CODE (Upwind PNS Solver)

RESULTS (Experimental & Computational)

- **Flow Visualization**
- **Surface Pressures**
- **Surface Convective Heat Transfer**
- **Pitot-Pressure Flow-Field Surveys**

CONCLUDING REMARKS

ALL-BODY HYPERSONIC TEST PROGRAM FOR CFD CODE VALIDATION

OBJECTIVE: Establish benchmark experimental data base for generic hypersonic vehicle shape for validation and/or calibration of advanced CFD computer codes

MOTIVATION: Need for extensive hypersonic data to fully validate CFD codes to be used for NASP & other hypersonic vehicles

APPROACH: Conduct comprehensive test program for generic all-body hypersonic aircraft model in Ames 3.5-ft Hypersonic Wind Tunnel to obtain pertinent surface and flow-field data over broad range of test conditions

ALL-BODY HYPERSONIC AIRCRAFT EXPERIMENT

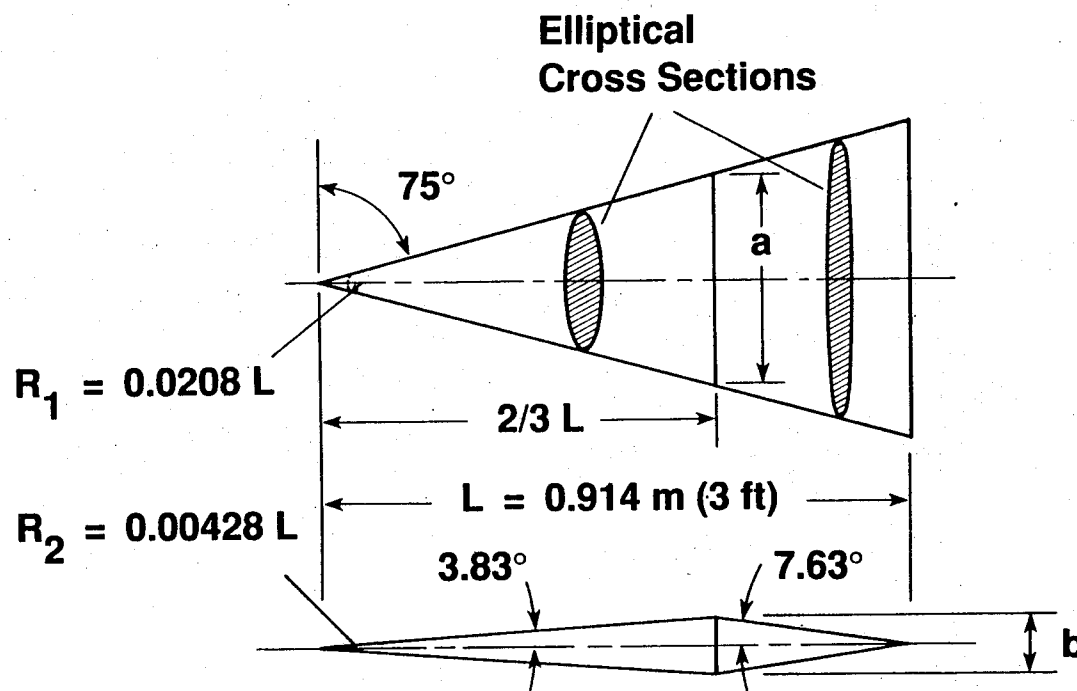
Ames 3.5-ft HWT

MODEL:

AMES GENERIC ALL-BODY HYPERSONIC MODEL:

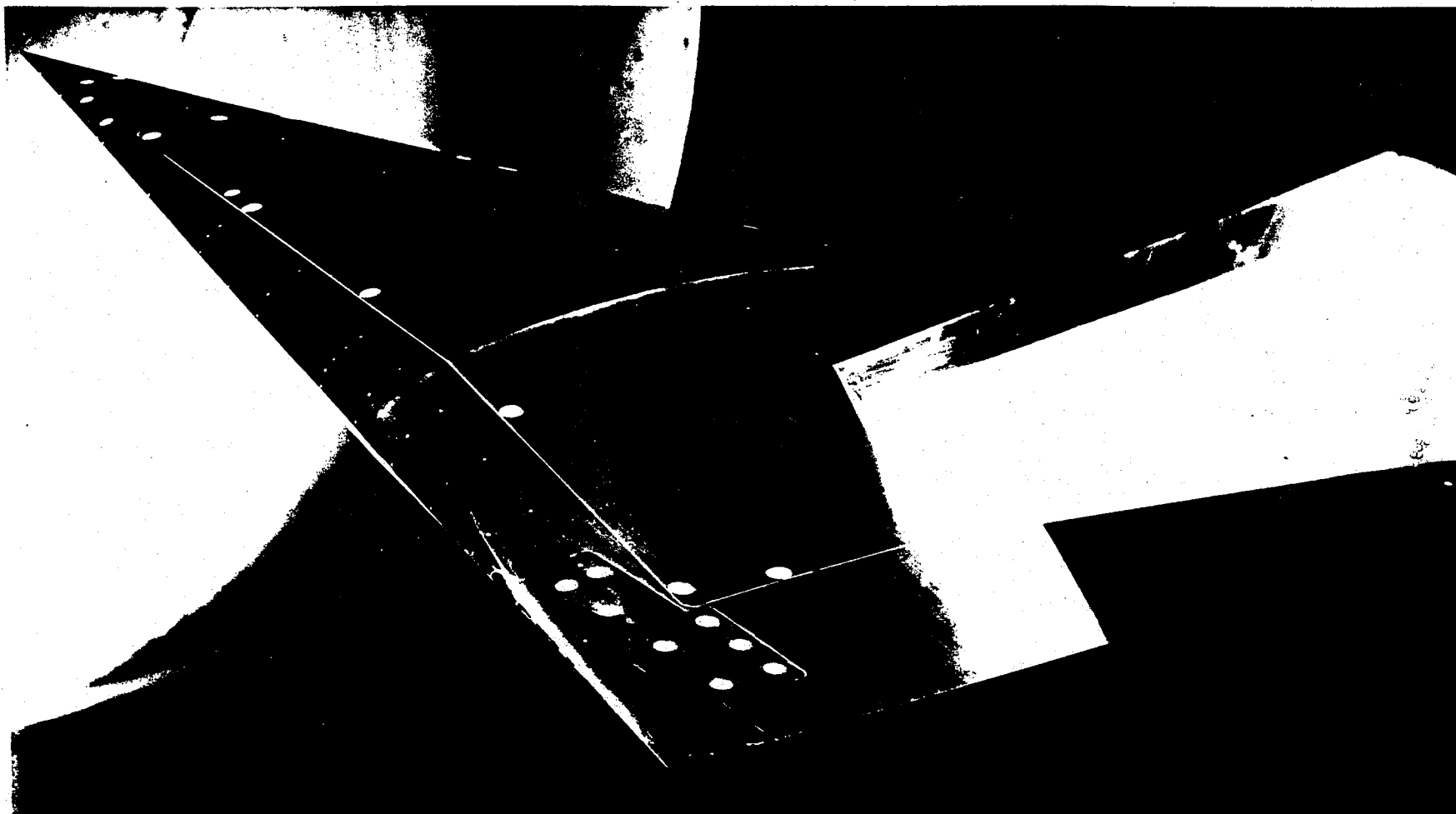
- **Delta Planform ($\Lambda = 75^\circ$)**
 - **Forebody — Elliptic Cone ($a/b = 4$)**
 - **Afterbody — Elliptic Cross Sections with Sharp Trailing Edge**
- **Sharp or Blunt Nose Tip**
- **With or Without Control Surfaces (Tested without to date)**
 - **Canard**
 - **Combination Horizontal/Vertical Tails**

ALL-BODY HYPERSONONIC AIRCRAFT MODEL W/O CONTROL SURFACES



Forebody – Elliptic Cone ($a/b = 4$) with Sharp or Blunt Nose Tip
Afterbody – Elliptical Cross Sections with Sharp Trailing Edge

**ALL-BODY HYPERSONIC AIRCRAFT MODEL
IN NASA/AMES 3.5 -FT HWT
W/O CONTROL SURFACES; LENGTH = 3 FT**



ALL-BODY HYPERSONIC AIRCRAFT EXPERIMENT

Ames 3.5-ft HWT

TEST CONDITIONS:

- $M_{\infty} = 5, 7, \text{ \& } 10$
- $Re_{\infty, L} = 1.5 \times 10^6 \text{ to } 25 \times 10^6$ (Laminar to Turbulent Flows)
- $\alpha = 0^{\circ} \text{ to } 15^{\circ}$ (Attached & Separated Flows)

MEASUREMENTS:

- Flow Visualization
 - Shadowgraphs
 - Surface Oil-Flow Patterns (Skin-Friction Lines)
- Surface Pressures
- Surface Convective Heat Transfer (Selected Areas)
- Flow-Field Surveys
 - Pitot-Pressure Probes
 - Laser Doppler Velocimetry (To be done)
 - Mean Velocities
 - Turbulence Quantities

UPS CODE

(Upwind PNS Solver)

CHARACTERISTICS OF ALGORITHM:

- **Second-order accurate and upwind in crossflow directions**
- **First-order accurate in streamwise (marching) direction**
- **Implicit**
- **Finite Volume**

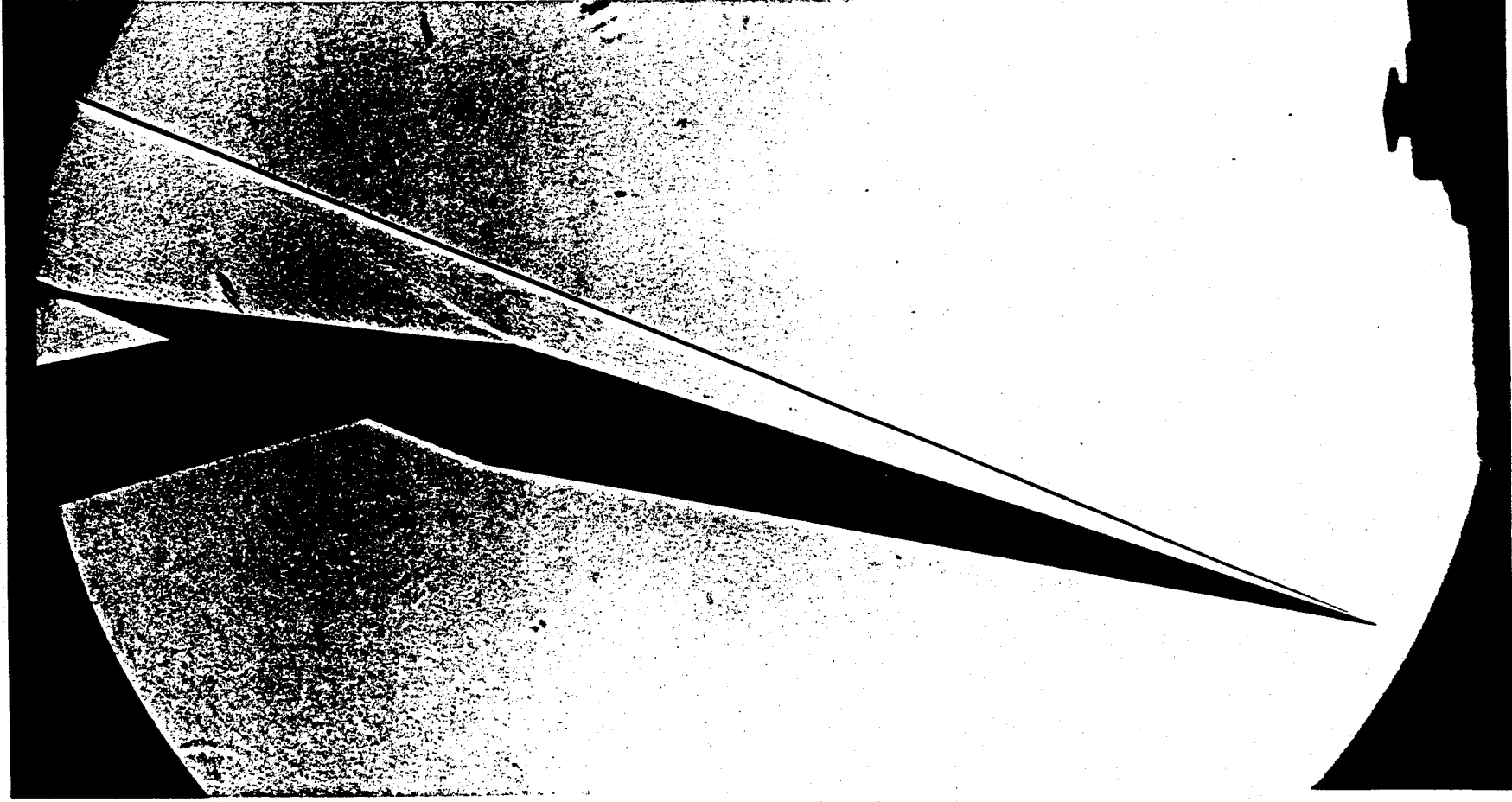
ADVANTAGES OF UPWIND SCHEMES:

- **Shock waves are captured sharply and without oscillation**
- **User specification of smoothing parameters is not required**

PRESENT ASSUMPTIONS:

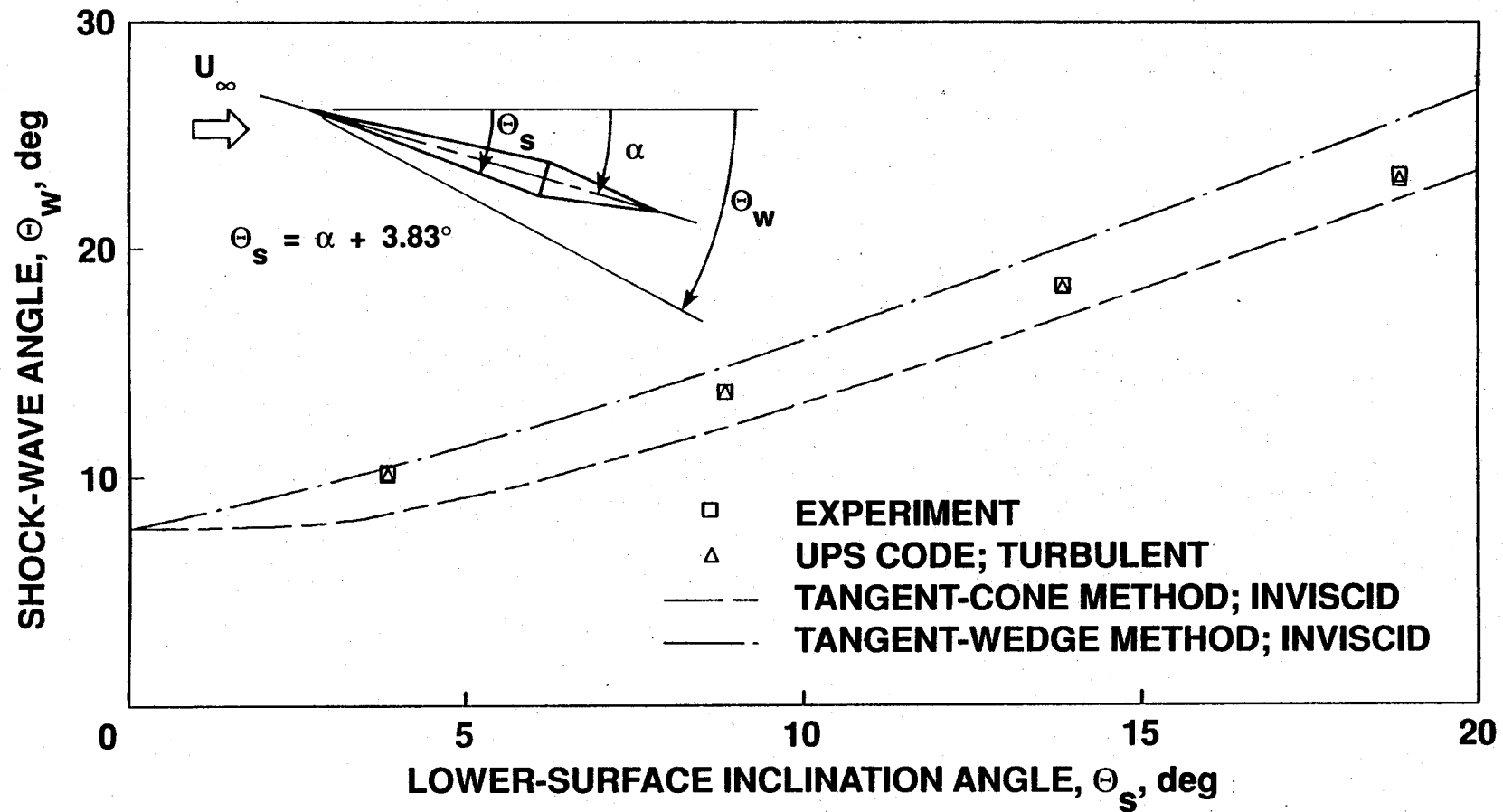
- **Laminar flow**
- **Turbulent flow**
 - **Boundary-layer transition specified**
 - **Baldwin-Lomax turbulence model**
- **Perfect gas (Used for this study)**
- **Equilibrium or nonequilibrium air**

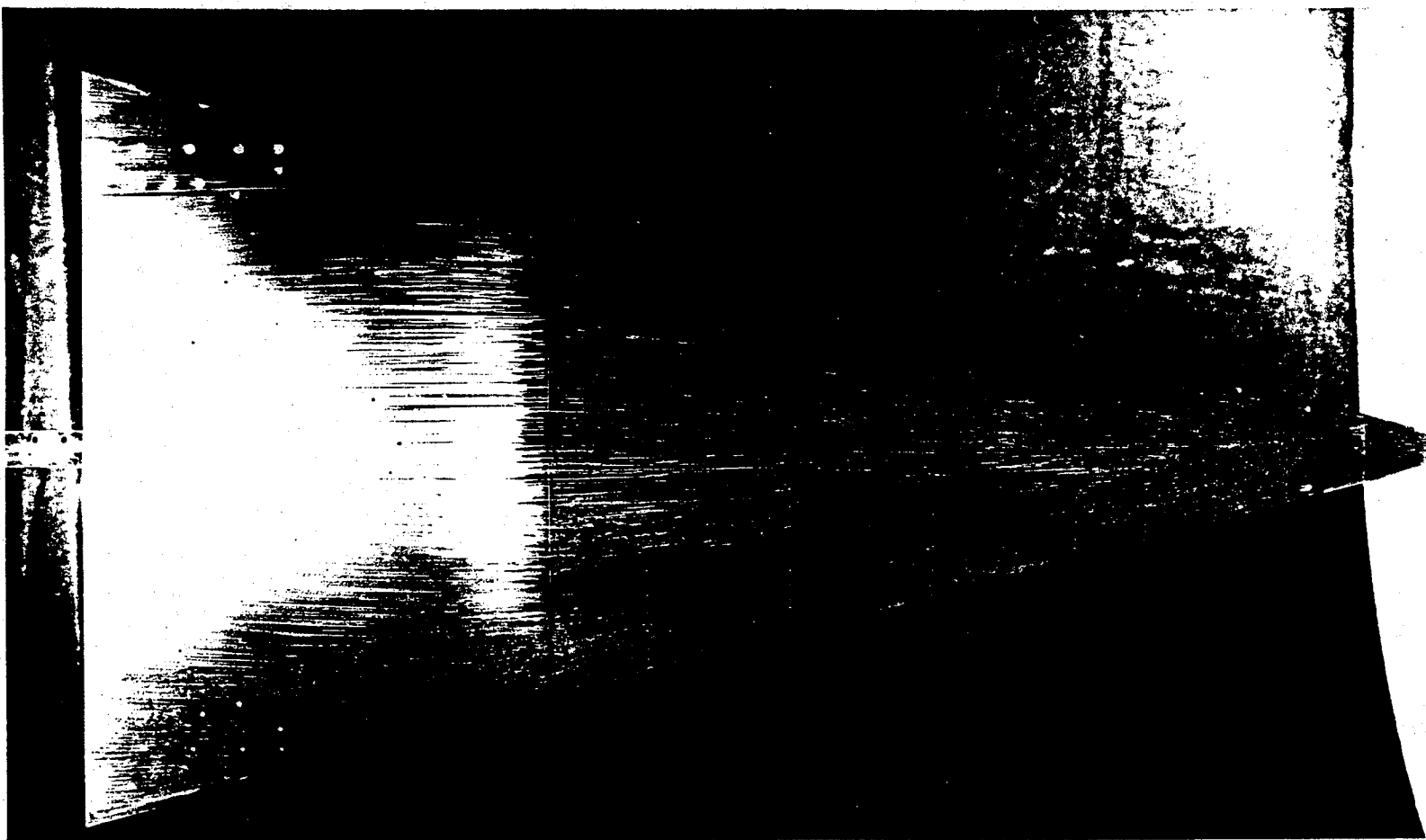
ALL-BODY MODEL SHADOWGRAPH
 $\alpha = 15^\circ; M_\infty = 7.4; Re_{L,\infty} = 15 \times 10^6$



FOREBODY SHOCK-WAVE ANGLE

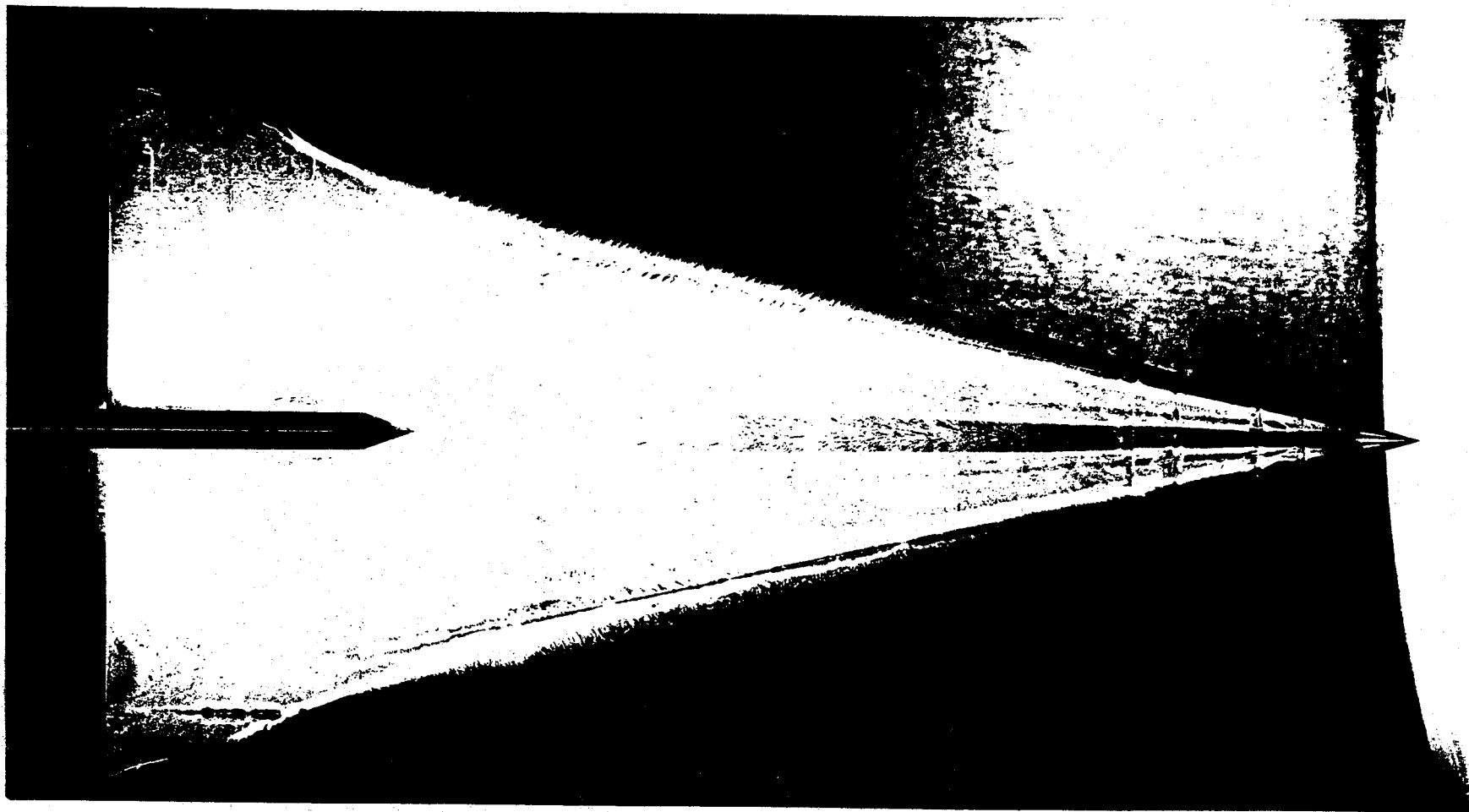
$$M_{\infty} = 7.4; Re_{\infty, L} = 15 \times 10^6$$





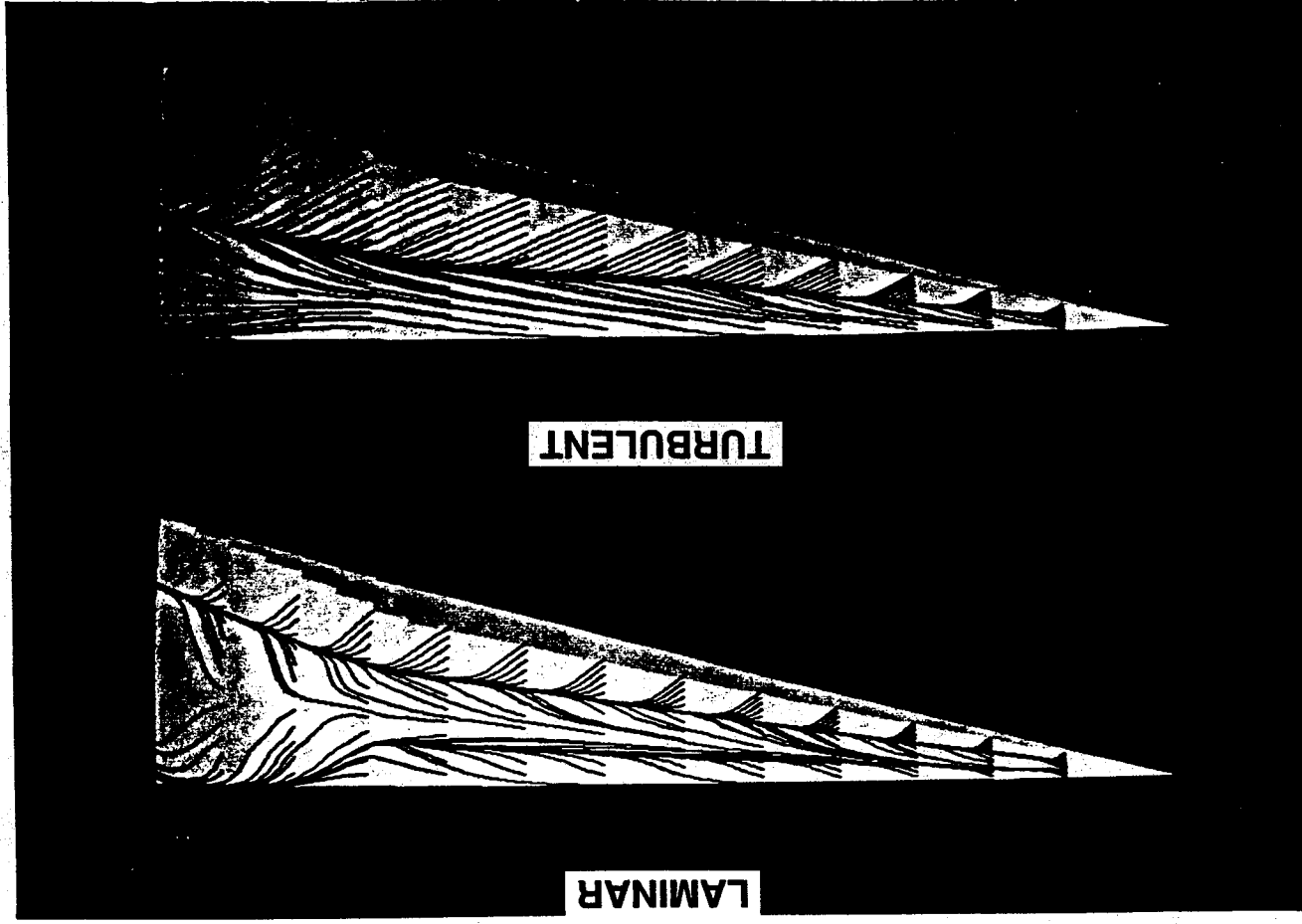
SURFACE OIL-FLOW PATTERN
 $\alpha = 15^\circ; M_\infty = 7.4; Re_{L,\infty} = 15 \times 10^6$
 WINDWARD

137



SURFACE OIL-FLOW PATTERN
 $\alpha = 15^\circ; M_\infty = 7.4; Re_{\infty, L} = 15 \times 10^6$
LEEWARD

PARTICLE TRACES FOR LEEWARD SURFACE BY UPS CODE
 (Simulated Surface Oil-flow Patterns; $x/L = 0.0$ to 0.9)
 $\alpha = 15^\circ; M_\infty = 7.4; Re_{\infty, L} = 15 \times 10^6$



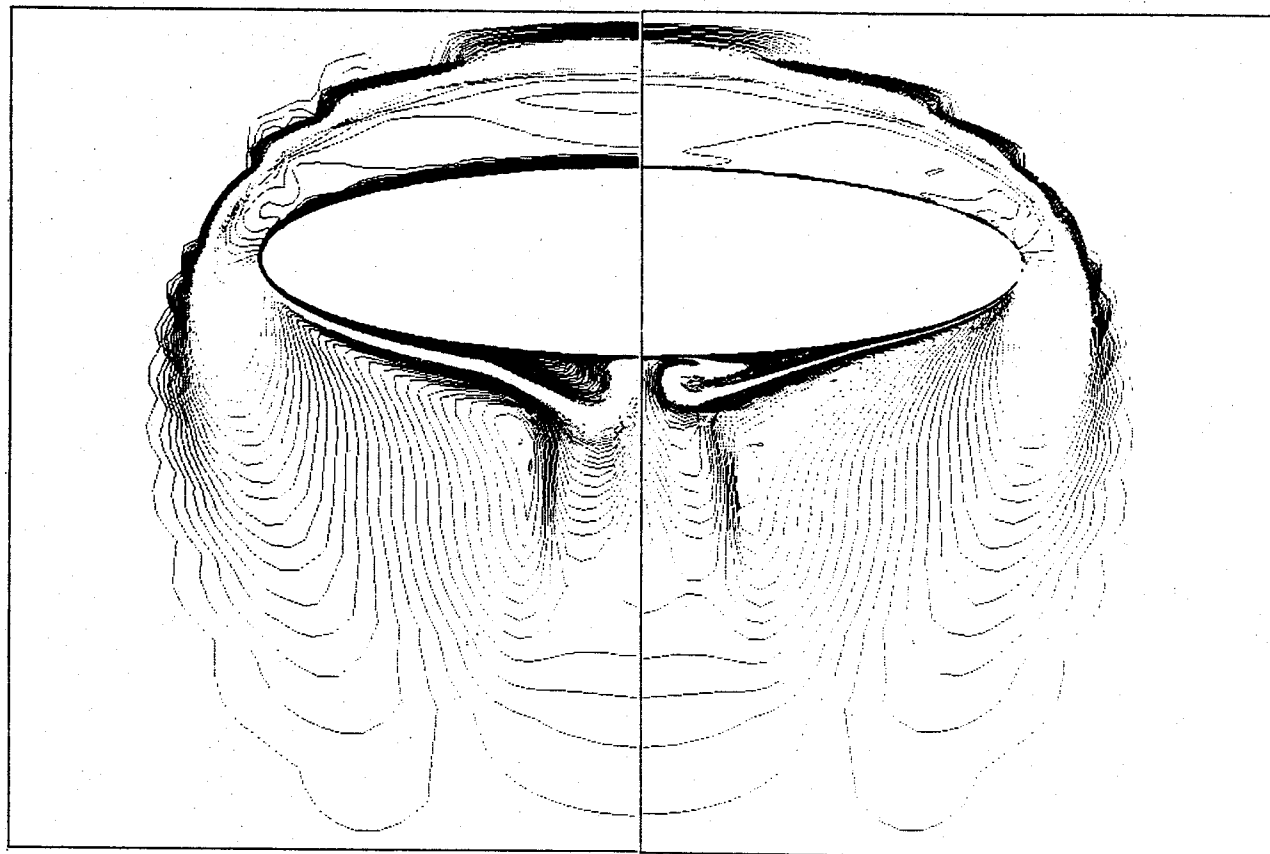
MACH NUMBER CONTOURS FOR CROSS-FLOW PLANE BY UPS CODE

$$x/L = 0.6$$

$$\alpha = 15^\circ; M_\infty = 7.4; Re_\infty L = 15 \times 10^6$$

TURBULENT

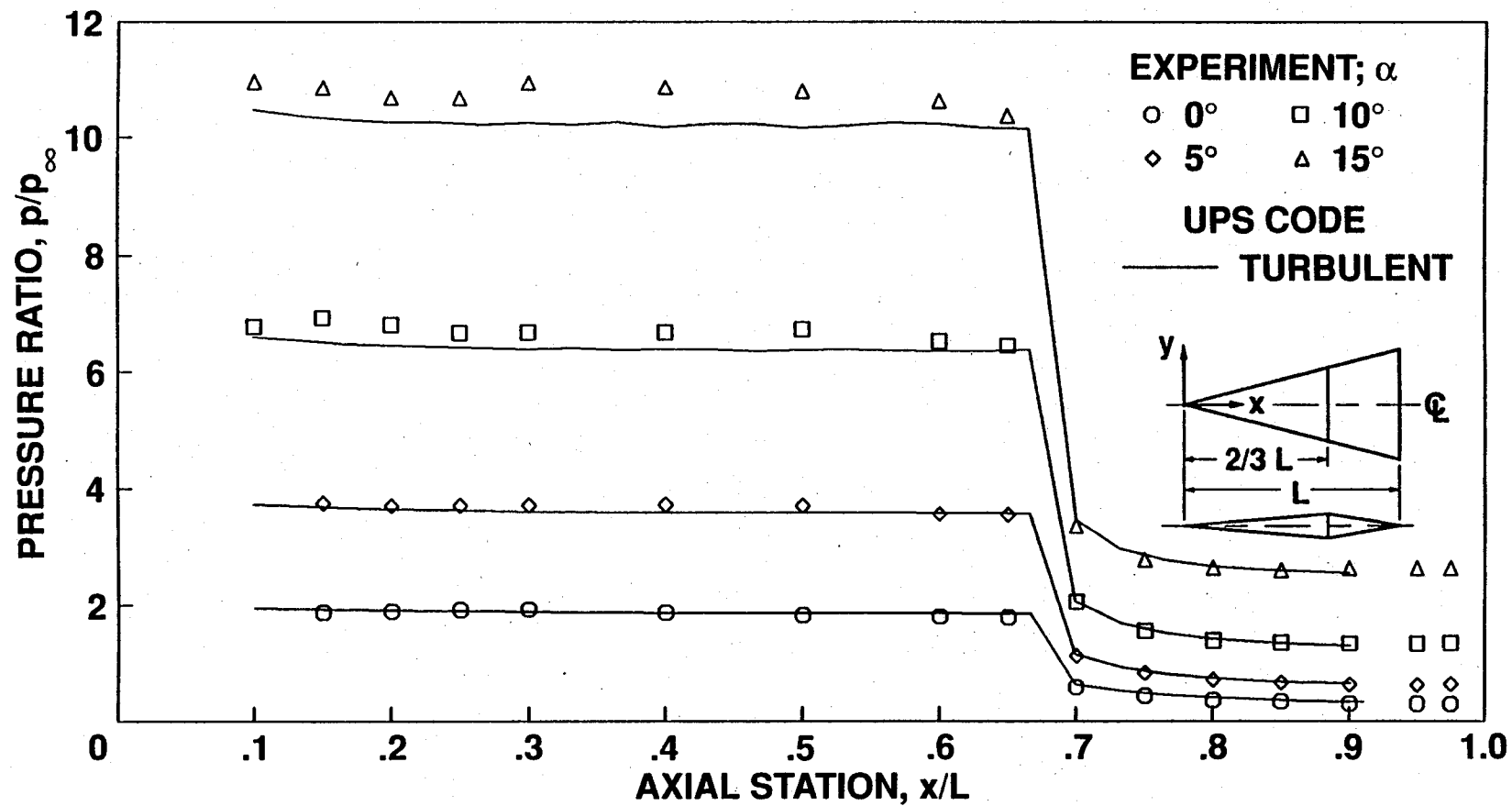
LAMINAR



EFFECT OF ANGLE OF ATTACK ON CENTERLINE SURFACE PRESSURES

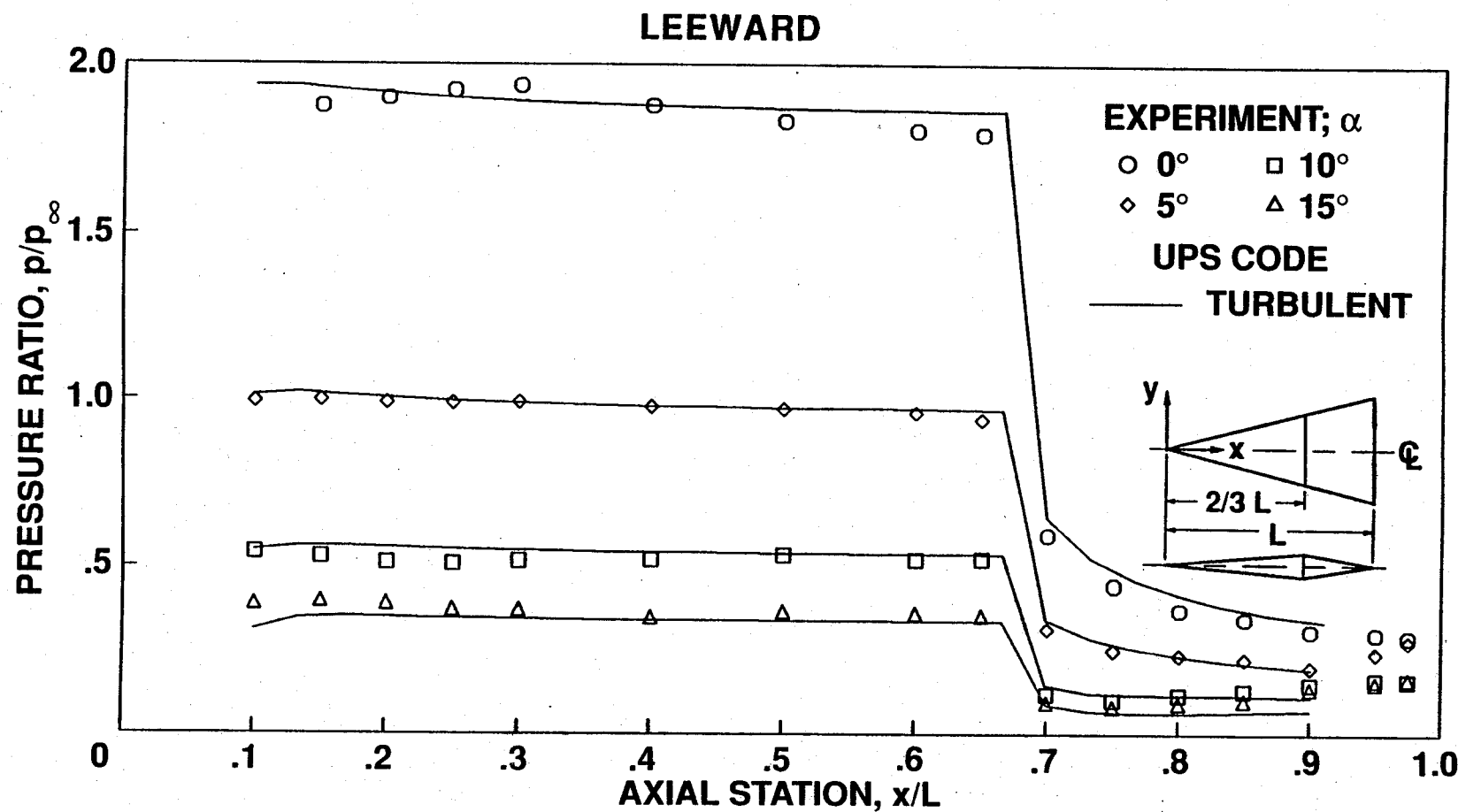
$$M_{\infty} = 7.4; Re_{\infty, L} = 15 \times 10^6$$

WINDWARD



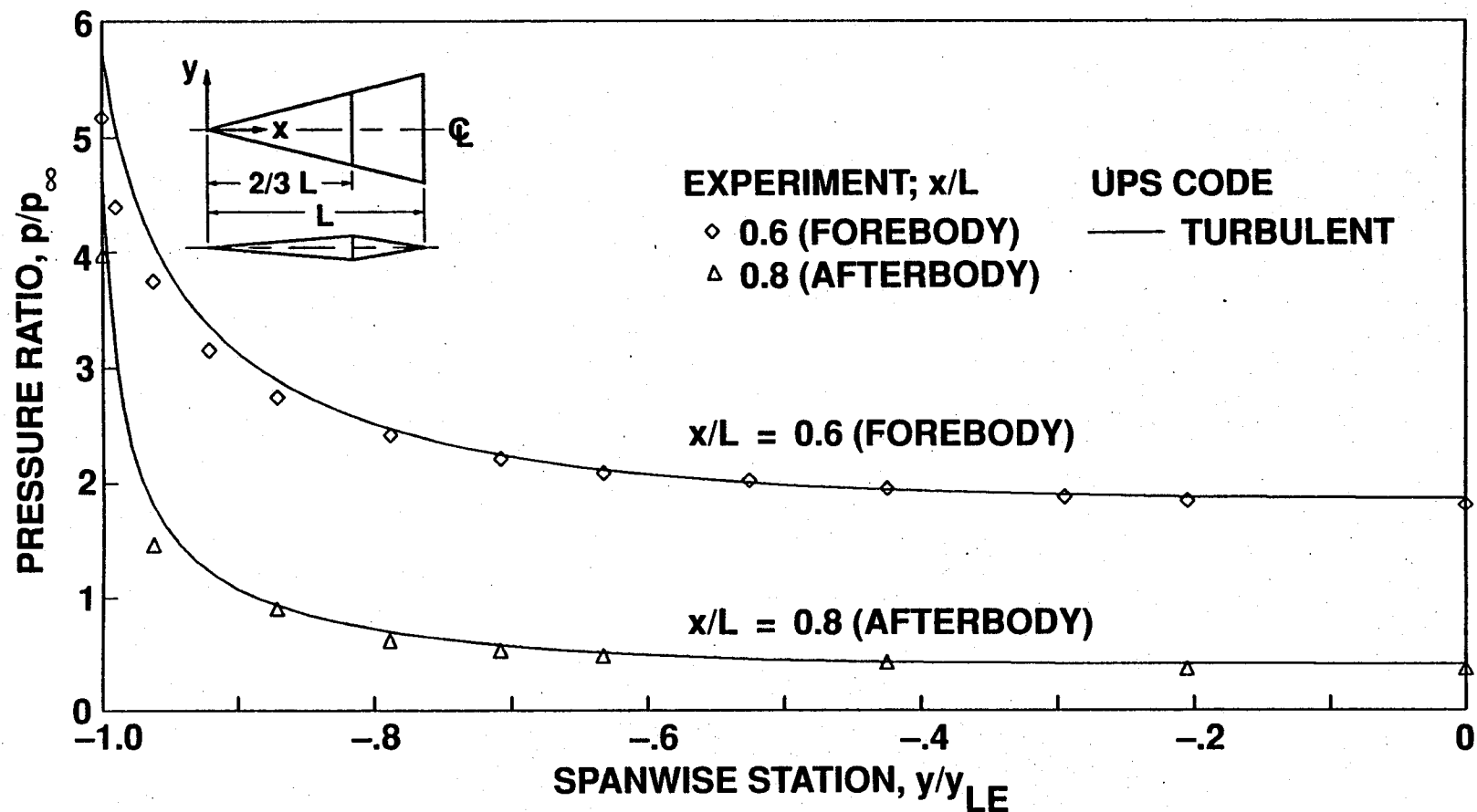
EFFECT OF ANGLE OF ATTACK ON CENTERLINE SURFACE PRESSURES

$$M_{\infty} = 7.4; Re_{\infty, L} = 15 \times 10^6$$



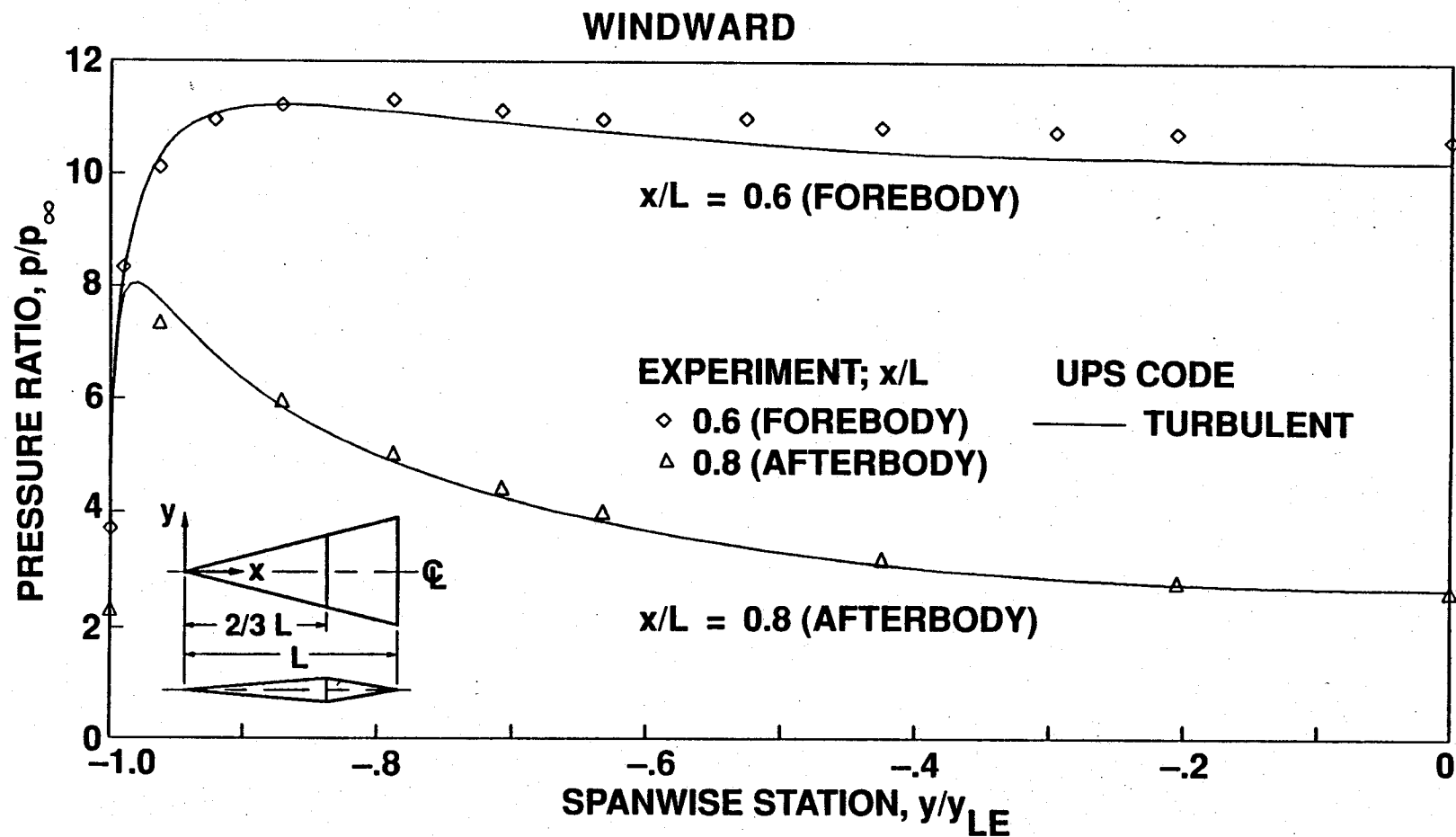
SPANWISE SURFACE-PRESSURE DISTRIBUTIONS FOR FOREBODY AND AFTERBODY

$\alpha = 0^\circ$; $M_\infty = 7.4$; $Re_{\infty, L} = 15 \times 10^6$



SPANWISE SURFACE-PRESSURE DISTRIBUTIONS FOR FOREBODY AND AFTERBODY

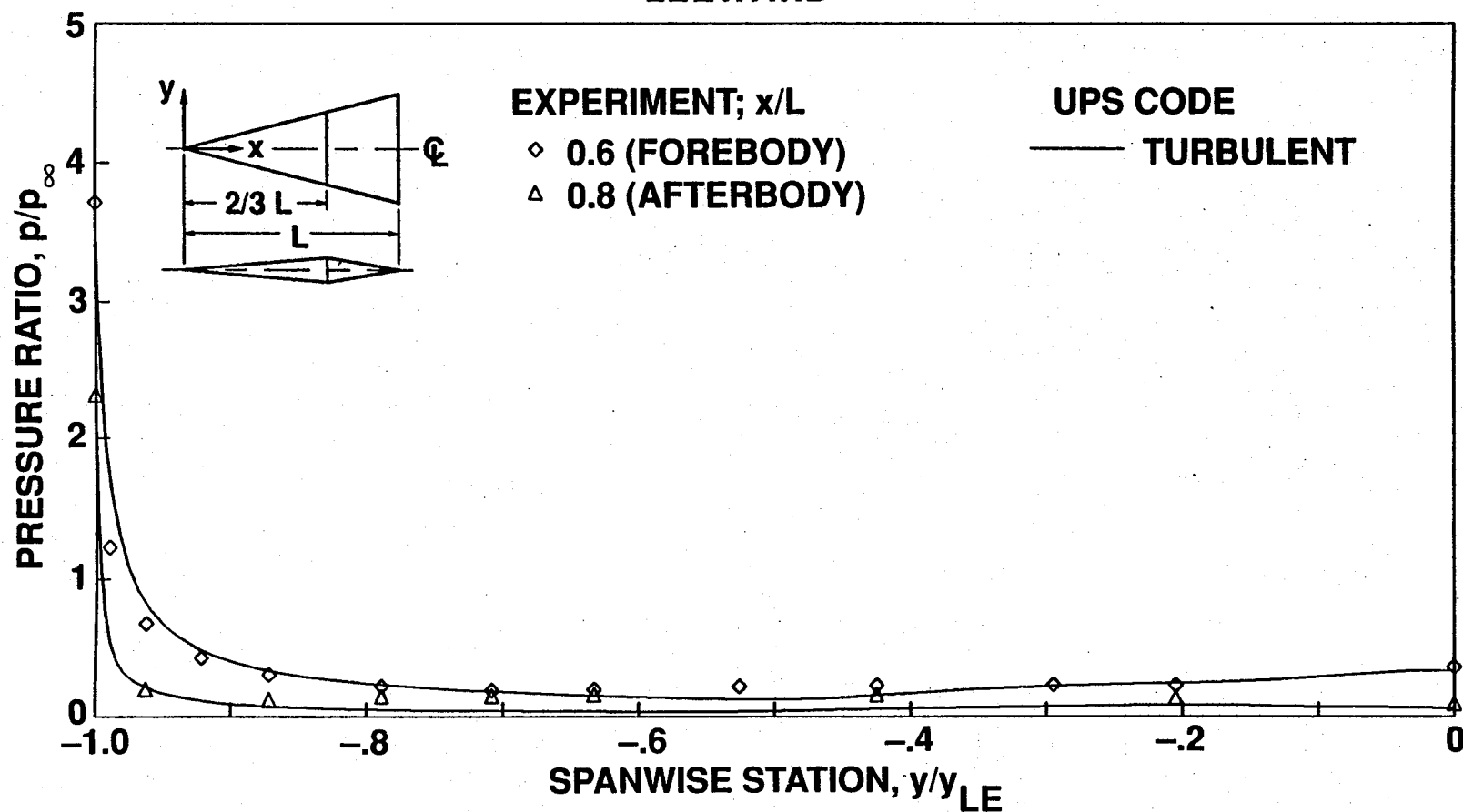
$$\alpha = 15^\circ; M_\infty = 7.4; Re_{\infty, L} = 15 \times 10^6$$



SPANWISE SURFACE-PRESSURE DISTRIBUTIONS FOR FOREBODY AND AFTERBODY

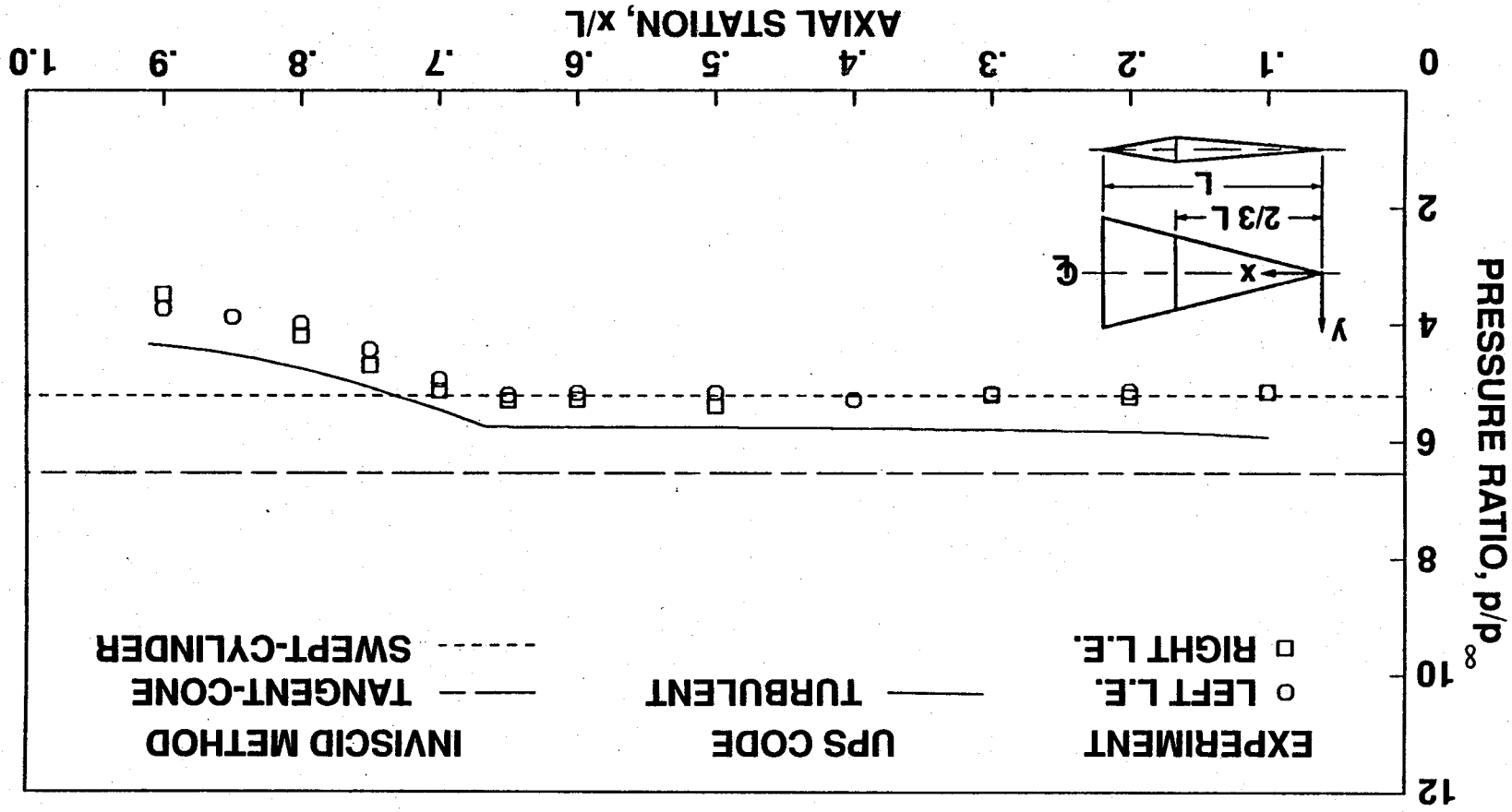
$\alpha = 15^\circ$; $M_\infty = 7.4$; $Re_{\infty,L} = 15 \times 10^6$

LEEWARD



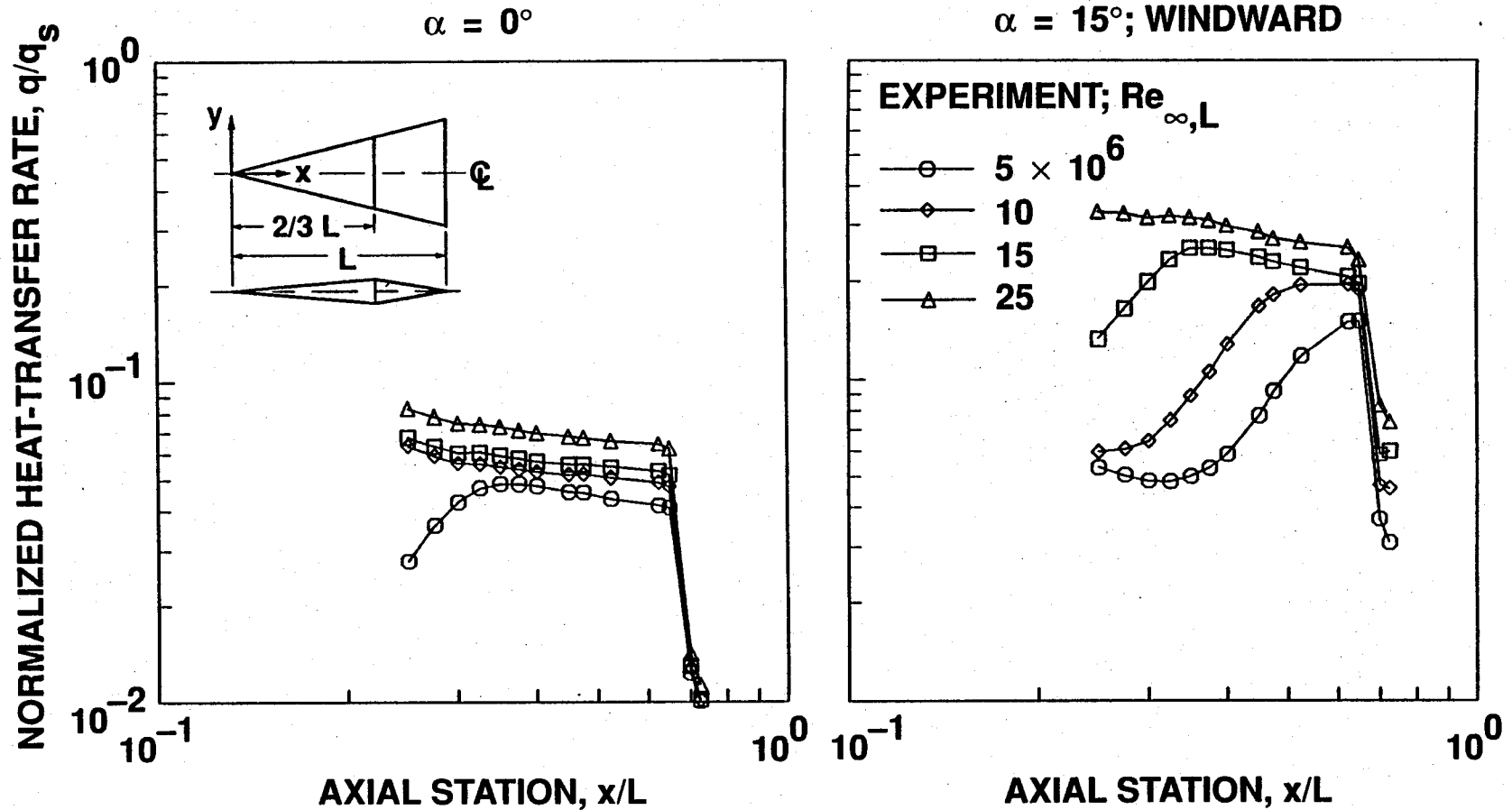
LEADING-EDGE SURFACE PRESSURES

$$\alpha = 0^\circ; M_\infty = 7.4; Re_{\infty, L} = 15 \times 10^6$$



EFFECT OF REYNOLDS NUMBER ON CENTERLINE HEAT TRANSFER

$$M_{\infty} = 7.4; H_w/H_t = 0.4$$

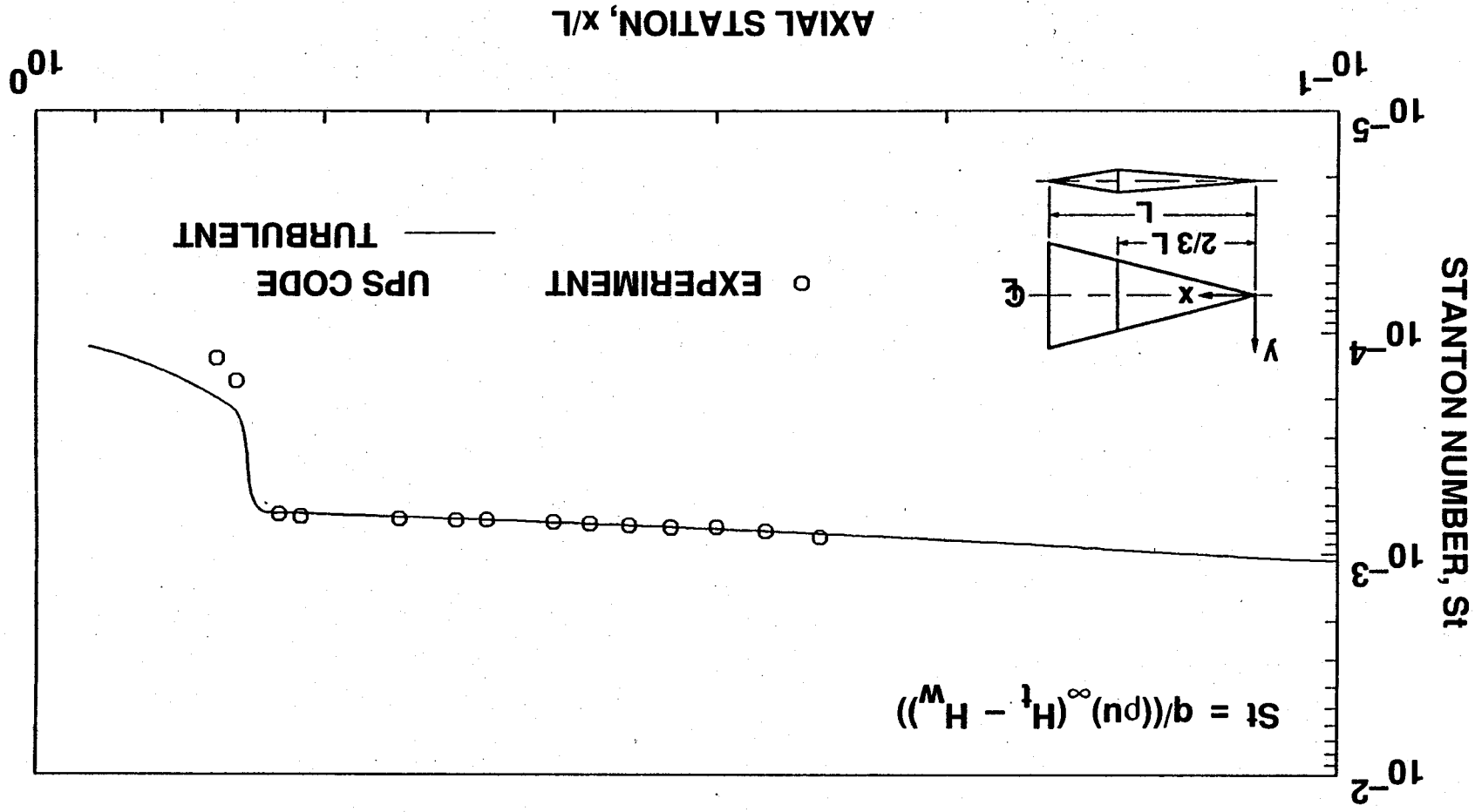


q_s = Stagnation-point heat-transfer rate for reference sphere ($R = 0.01L$)

CENTERLINE HEAT TRANSFER

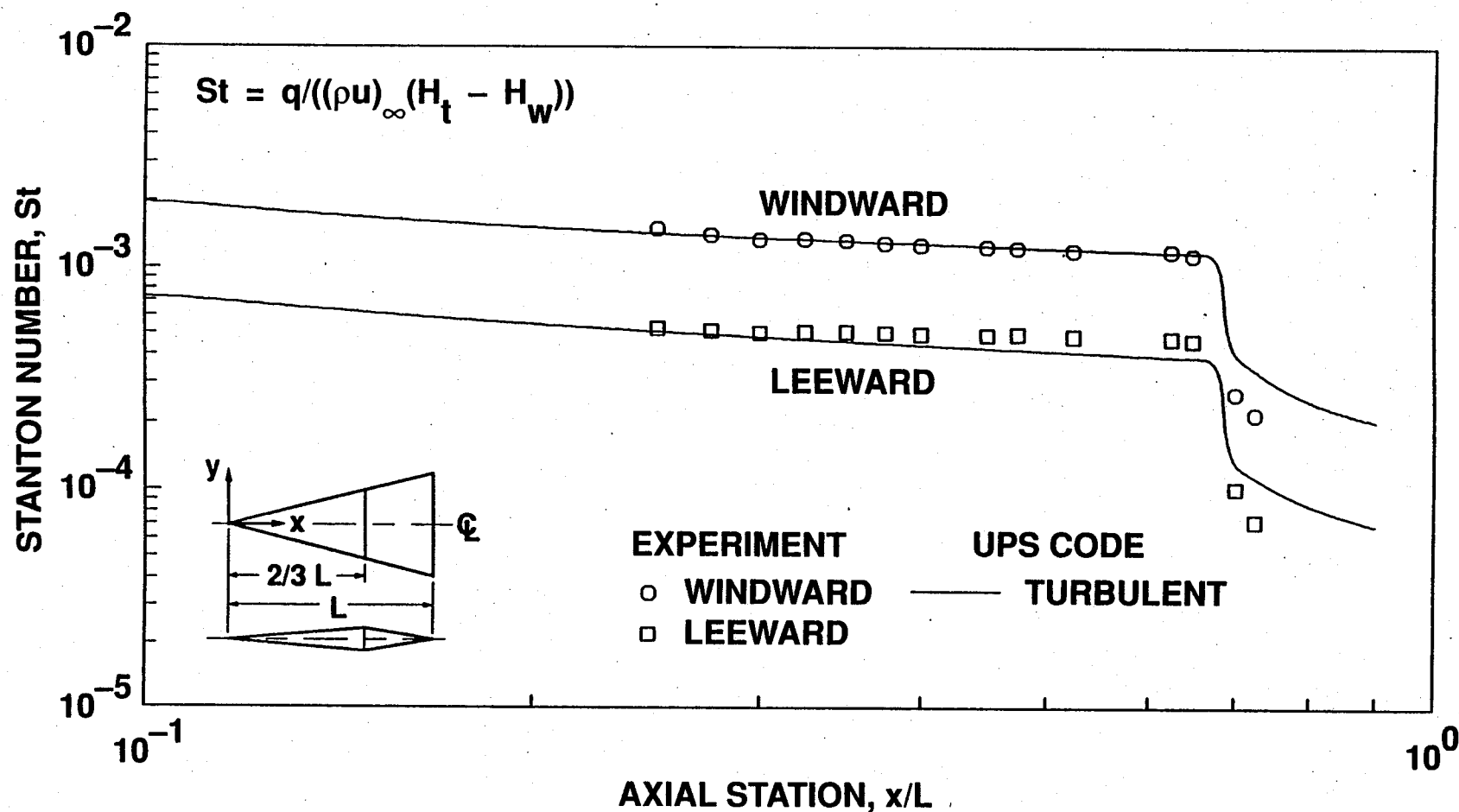
$\alpha = 0^\circ; M_\infty = 7.4; Re_{\infty, L} = 15 \times 10^6; H_w/H_t = 0.4$

$$St = q / ((\rho u)^\infty (H_t^\infty - H_w))$$



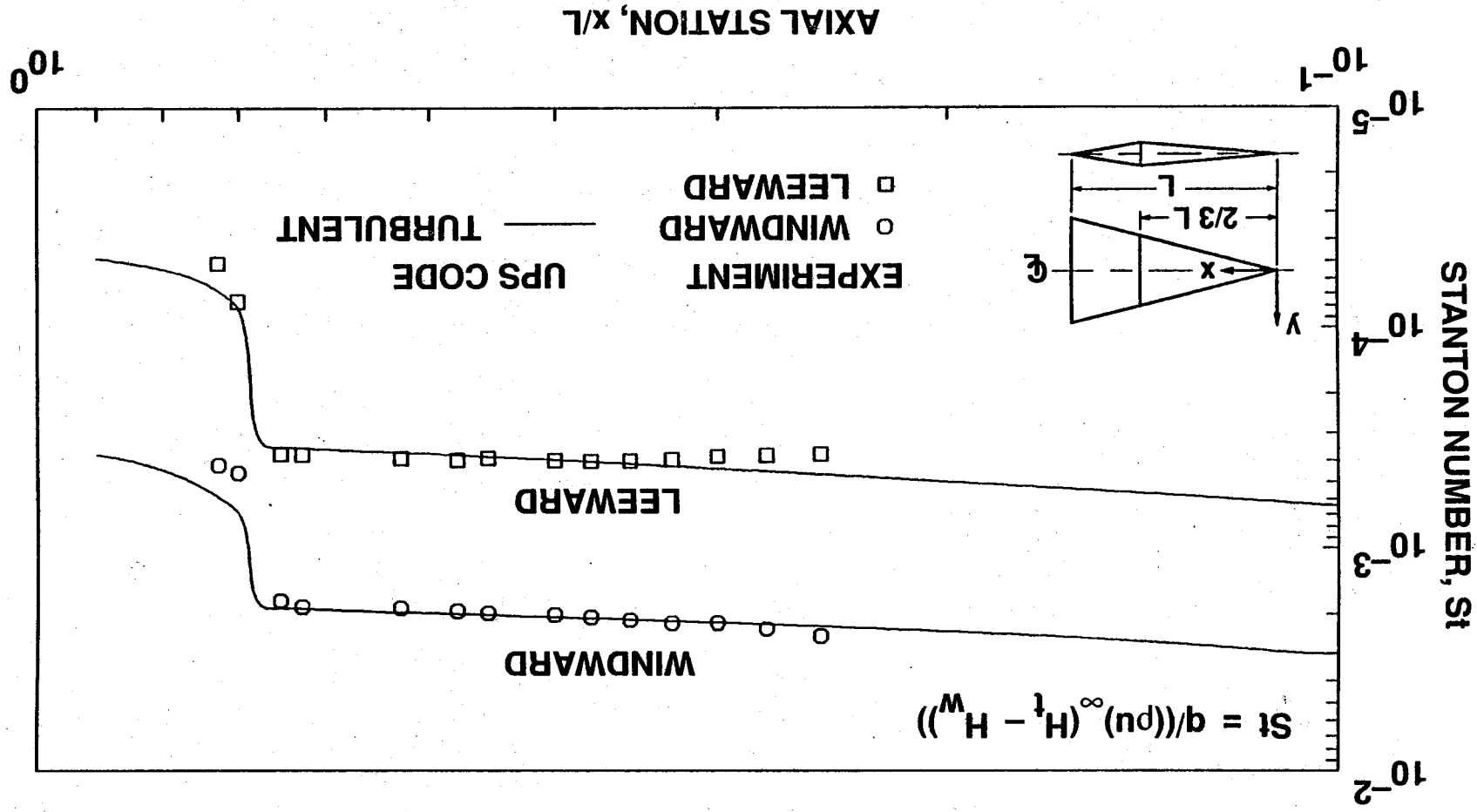
CENTERLINE HEAT TRANSFER

$$\alpha = 5^\circ; M_\infty = 7.4; Re_{\infty, L} = 15 \times 10^6; H_w/H_t = 0.4$$



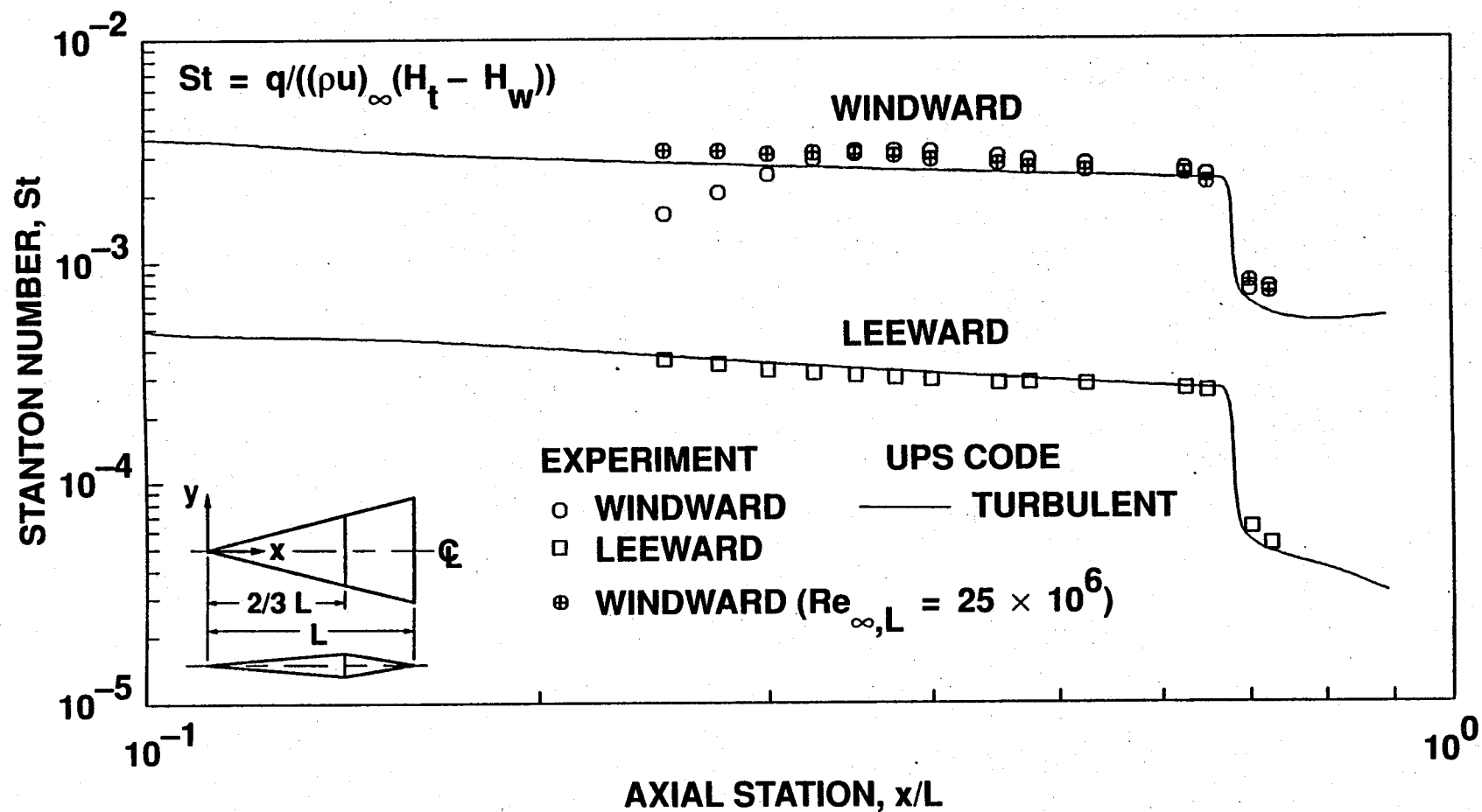
CENTERLINE HEAT TRANSFER

$\alpha = 10^\circ; M_\infty = 7.4; Re_{\infty, L} = 15 \times 10^6; H^w/H_t = 0.4$



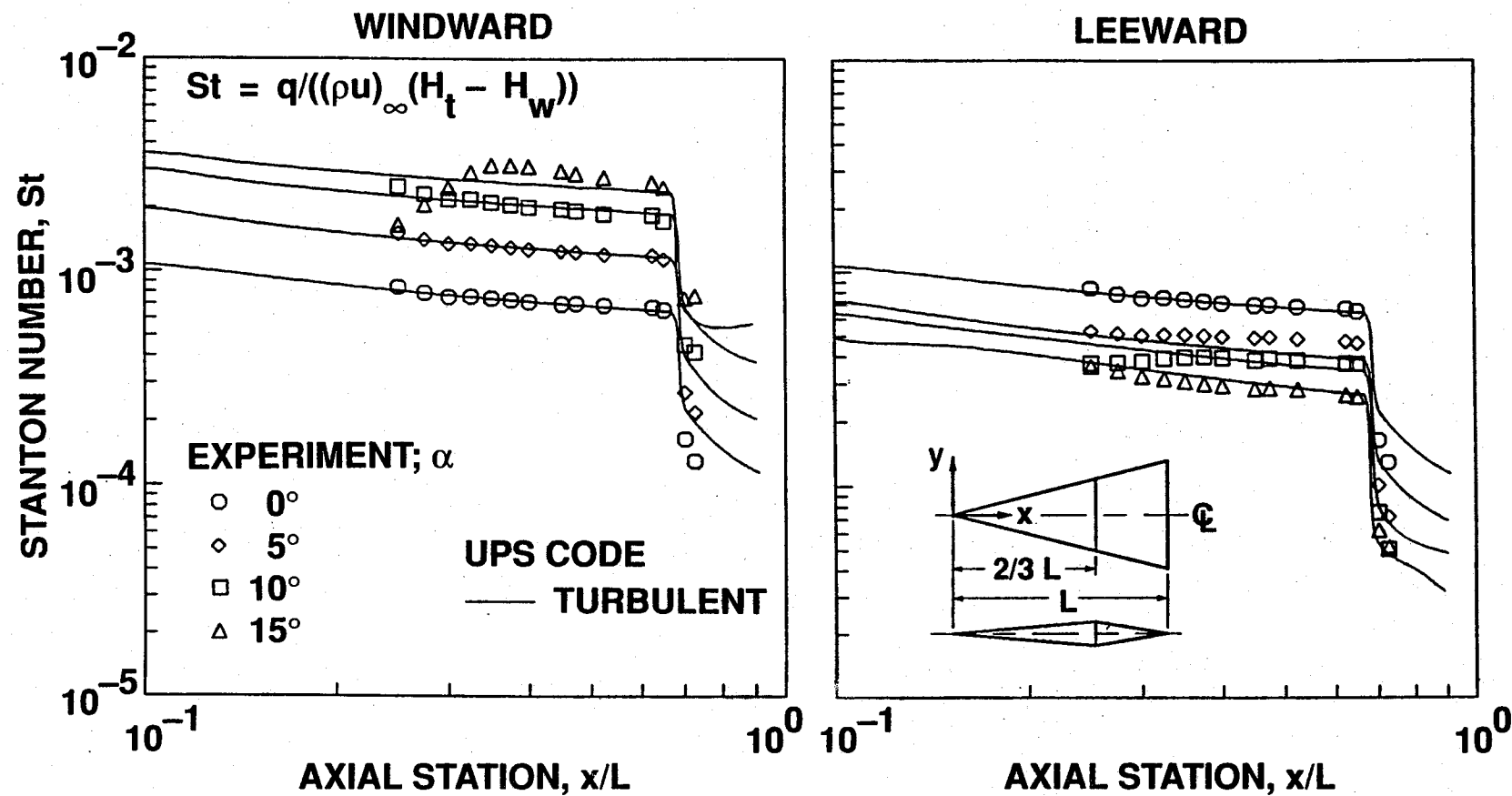
CENTERLINE HEAT TRANSFER

$$\alpha = 15^\circ; M_\infty = 7.4; Re_{\infty, L} = 15 \times 10^6; H_w/H_t = 0.4$$



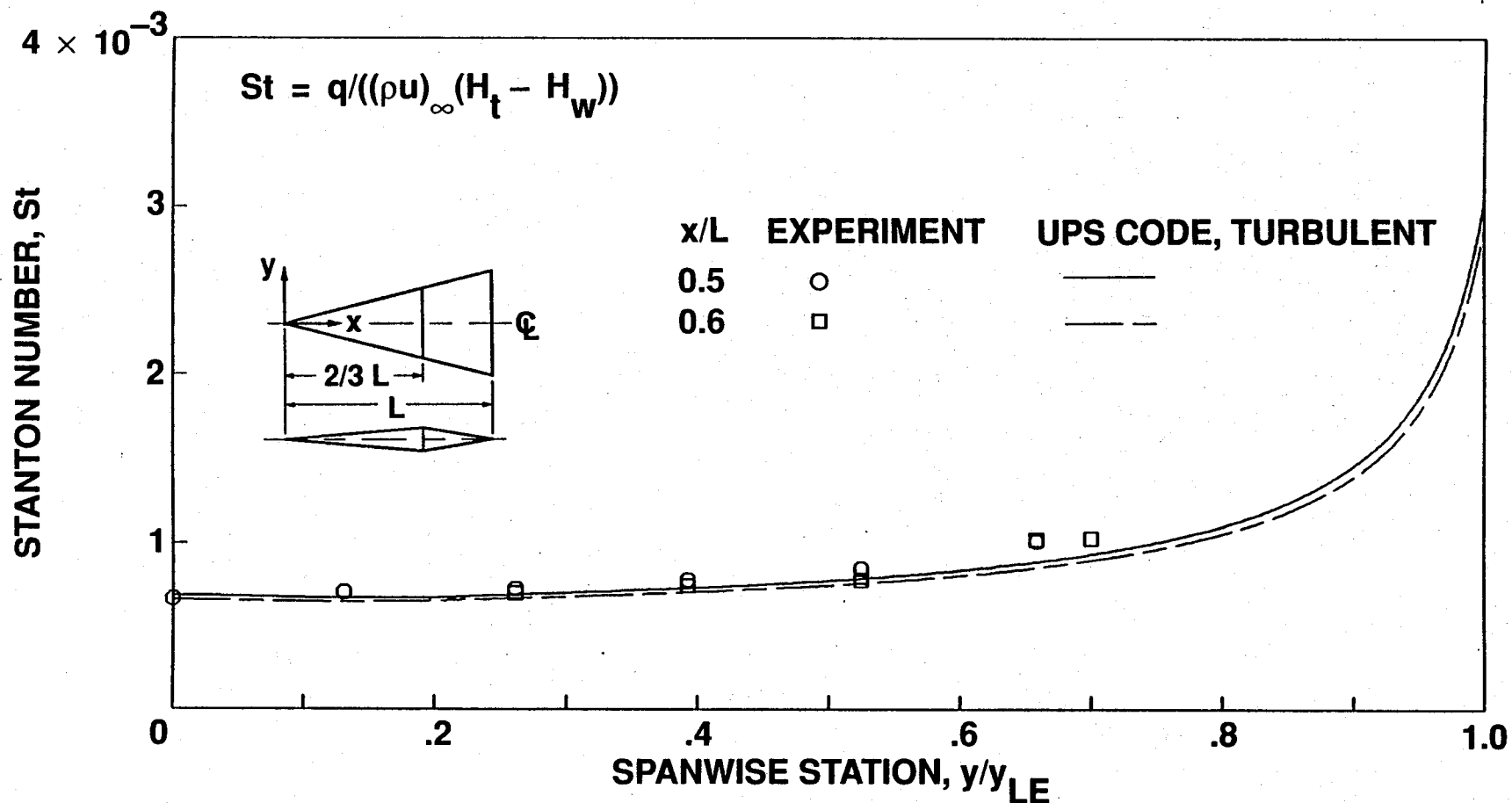
EFFECT OF ANGLE OF ATTACK ON CENTERLINE HEAT TRANSFER

$M_\infty = 7.4$; $Re_{\infty,L} = 15 \times 10^6$; $H_w/H_t = 0.4$



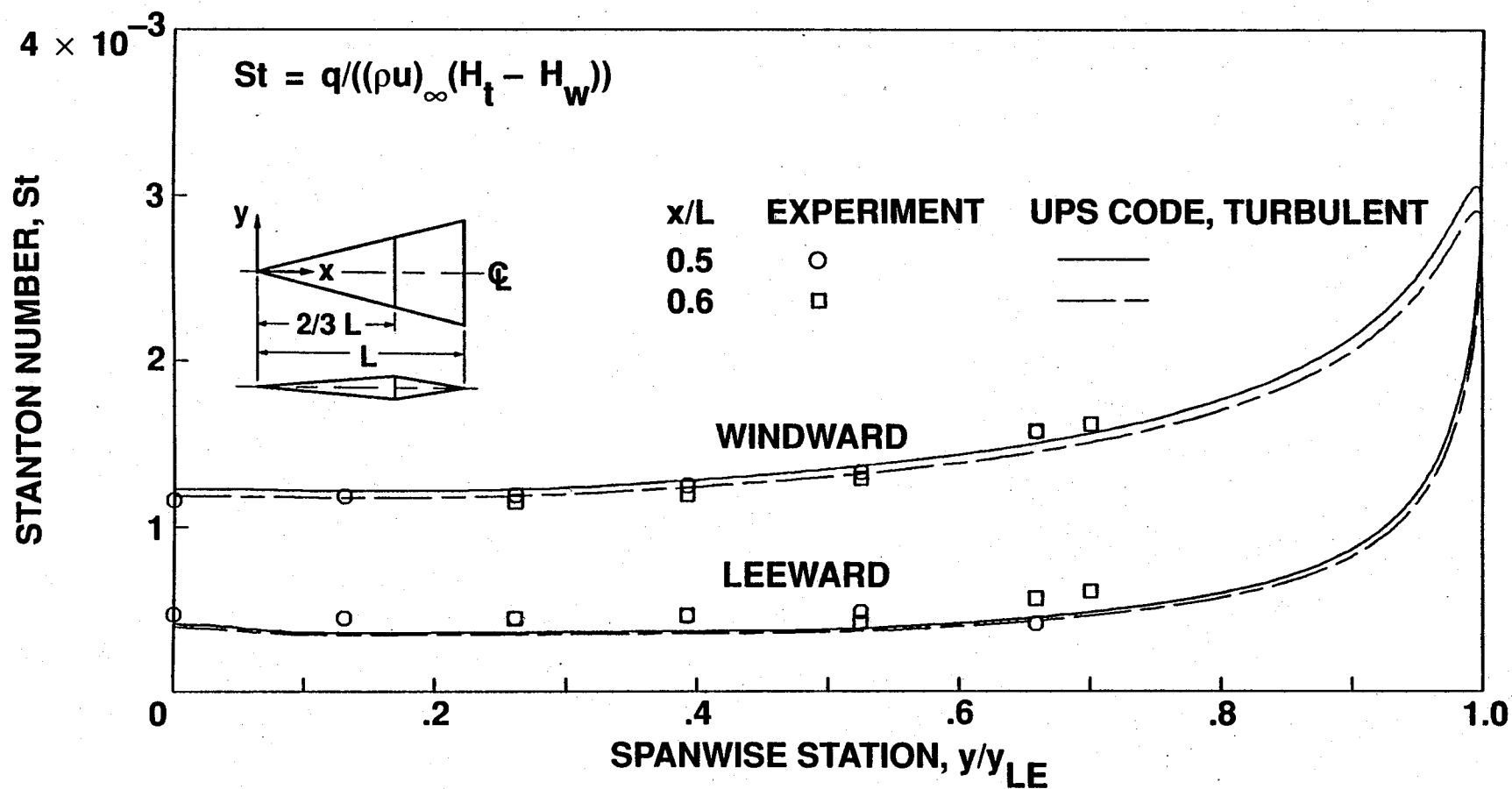
SPANWISE HEAT-TRANSFER DISTRIBUTIONS FOR FOREBODY

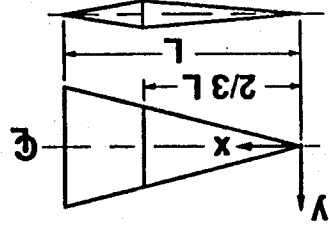
$\alpha = 0^\circ$; $M_\infty = 7.4$; $Re_{\infty,L} = 15 \times 10^6$; $H_w/H_t = 0.4$



SPANWISE HEAT-TRANSFER DISTRIBUTIONS FOR FOREBODY

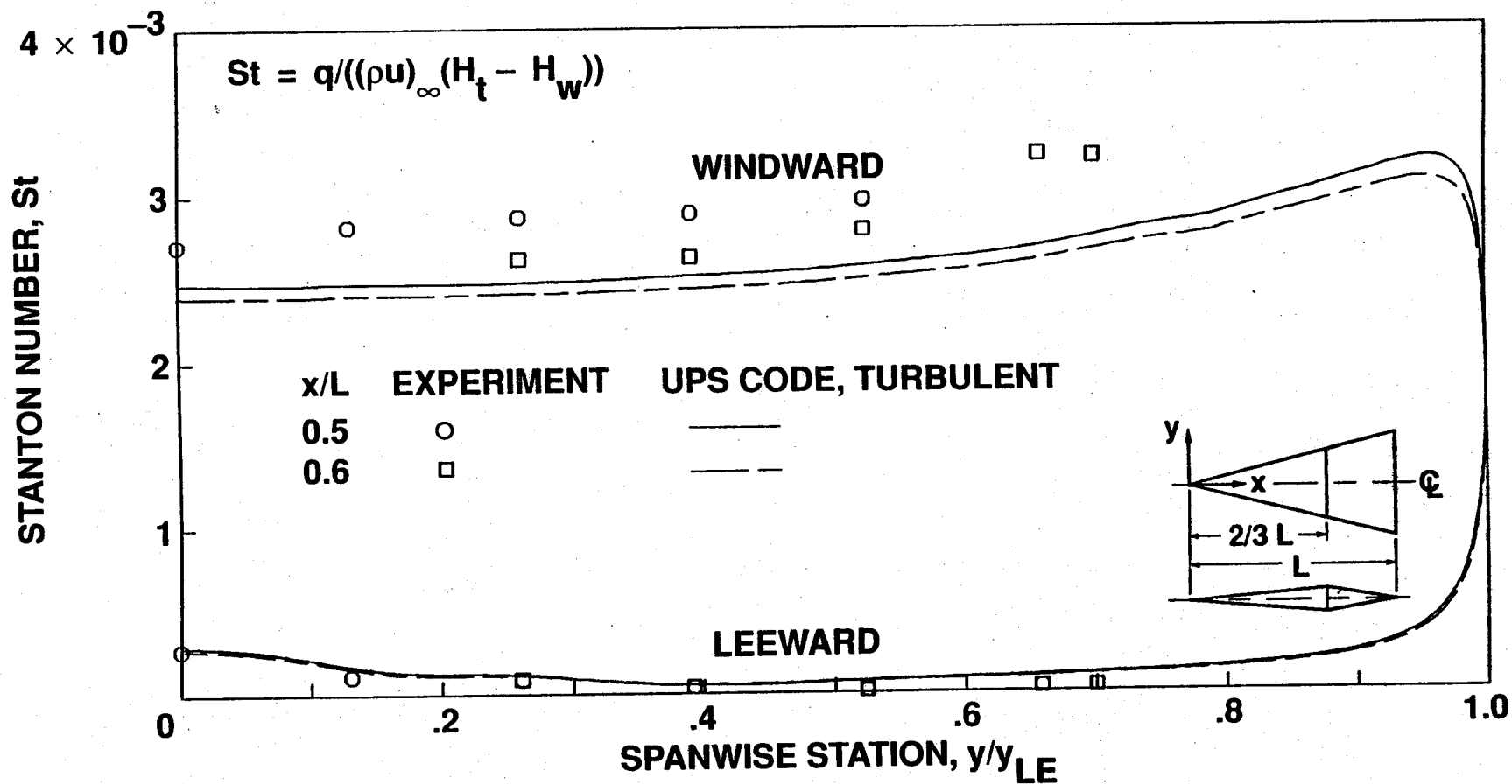
$\alpha = 5^\circ$; $M_\infty = 7.4$; $Re_{\infty,L} = 15 \times 10^6$; $H_w/H_t = 0.4$



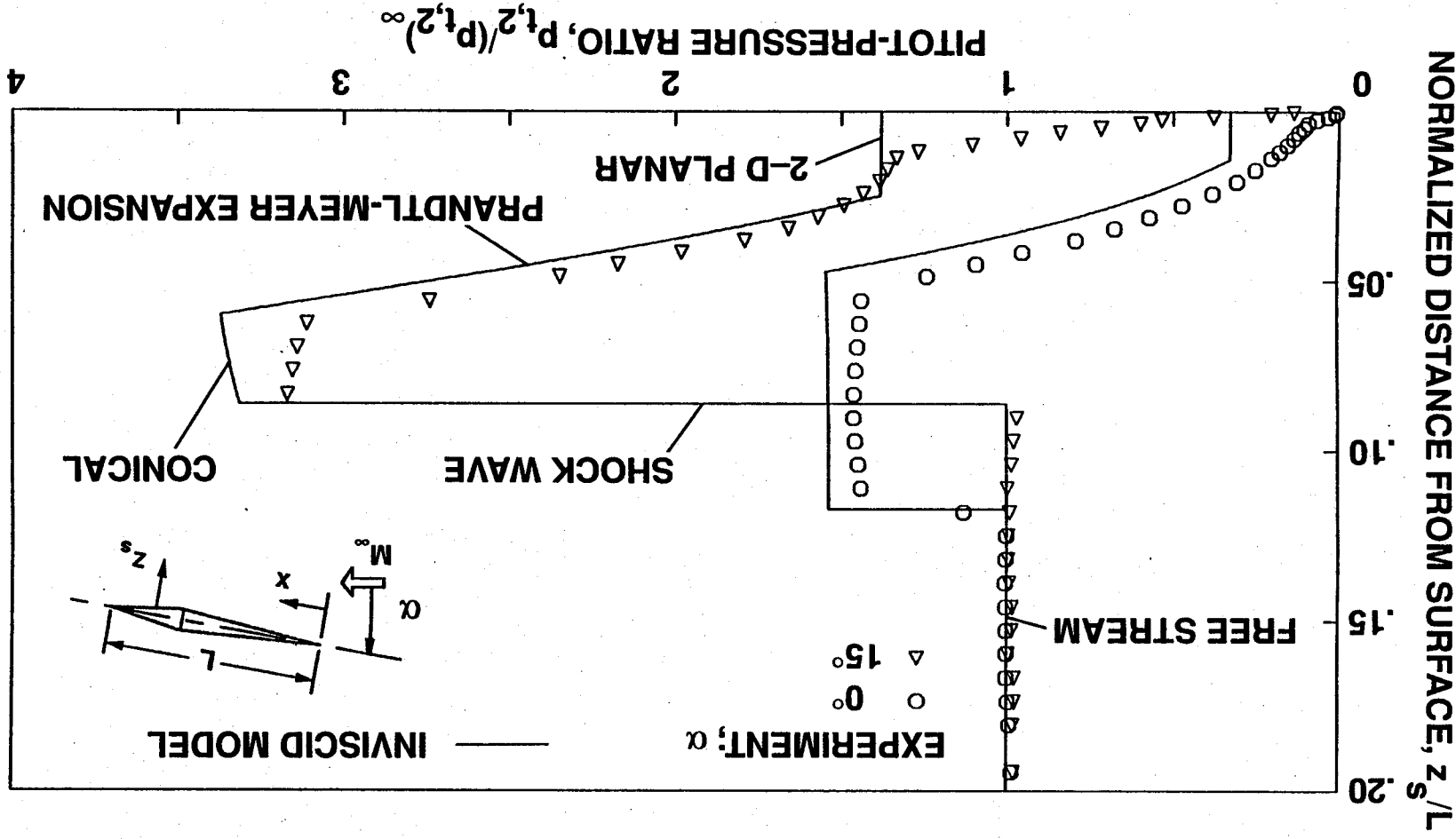
$$\alpha = 10^\circ; M_\infty = 7.4; Re_{\infty, L} = 15 \times 10^6; H_w/H_t = 0.4$$


SPANWISE HEAT-TRANSFER DISTRIBUTIONS FOR FOREBODY

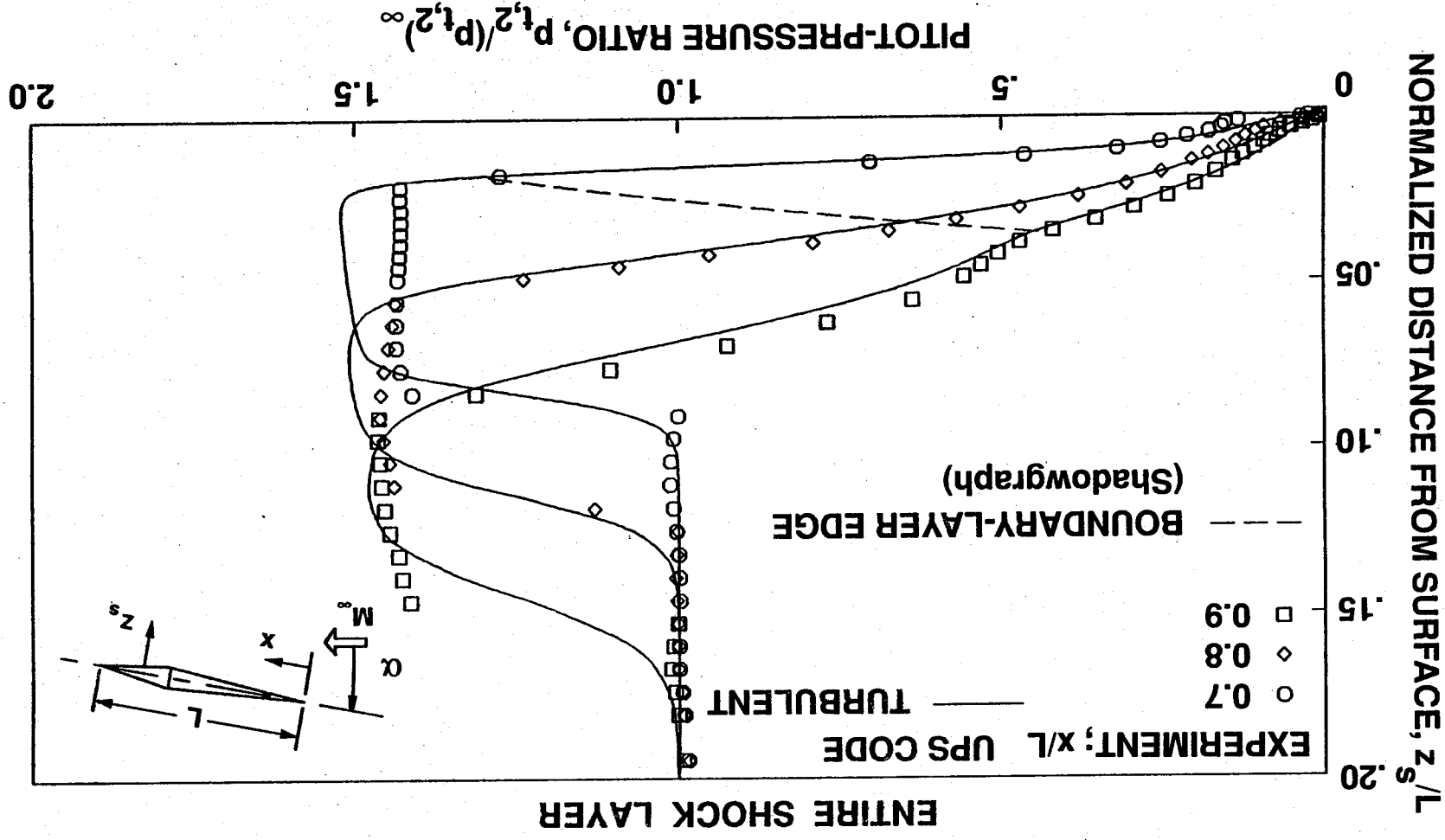
$$\alpha = 15^\circ; M_\infty = 7.4; Re_{\infty, L} = 15 \times 10^6; H_w/H_t = 0.4$$



COMPARISONS OF EXPERIMENTAL PITOT-PRESSURE DISTRIBUTIONS WITH INVISCID MODEL OF FLOW Afterbody Centerline; $x/L = 0.8$; $M_\infty = 7.4$; $Re_\infty L = 15 \times 10^6$

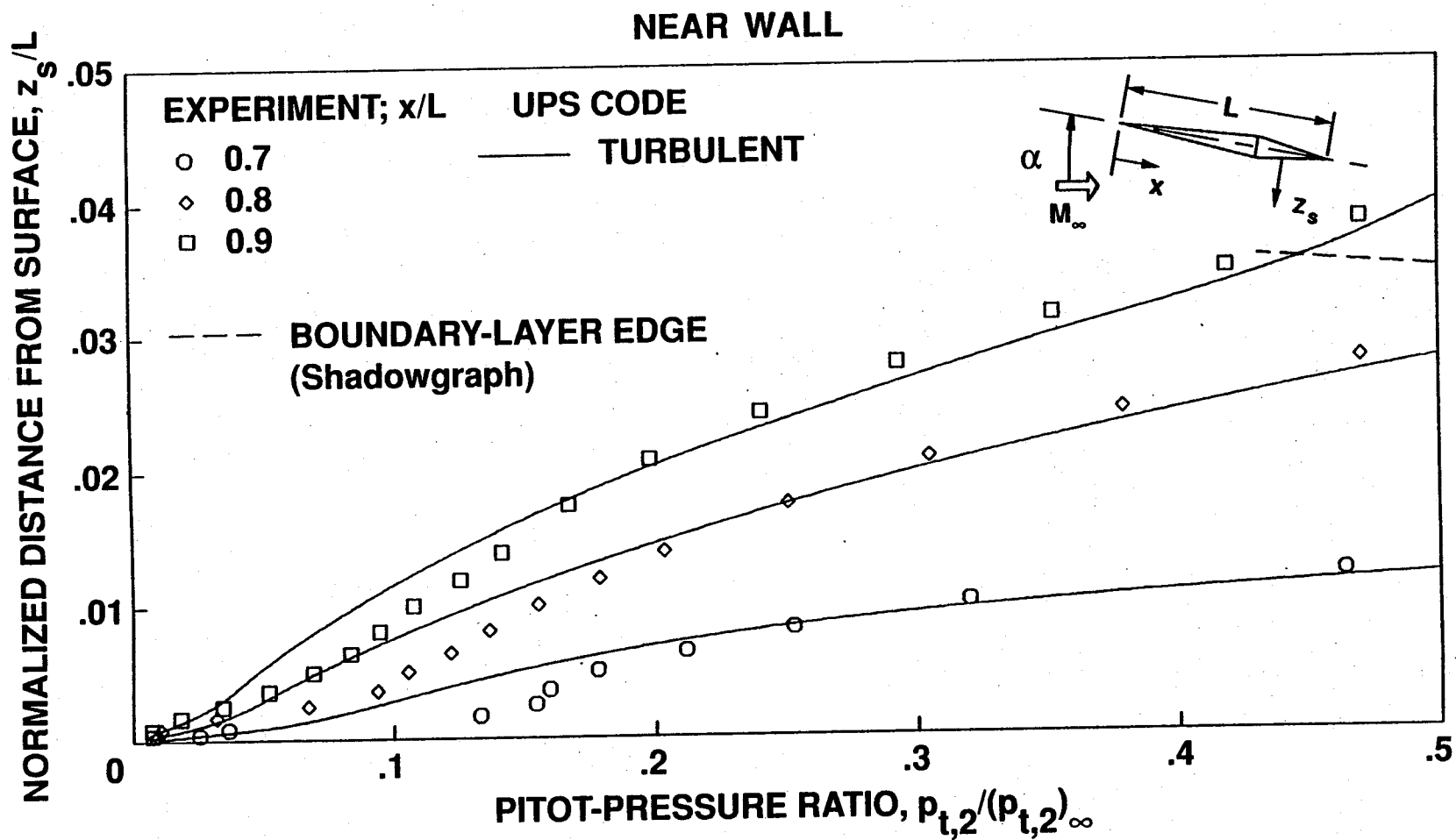


EXPERIMENTAL AND COMPUTATIONAL PITOT-PRESSURE DISTRIBUTIONS AT VARIOUS AFTERBODY STATIONS Centerline; $\alpha = 0^\circ$; $M_\infty = 7.4$; $Re_\infty L = 15 \times 10^6$

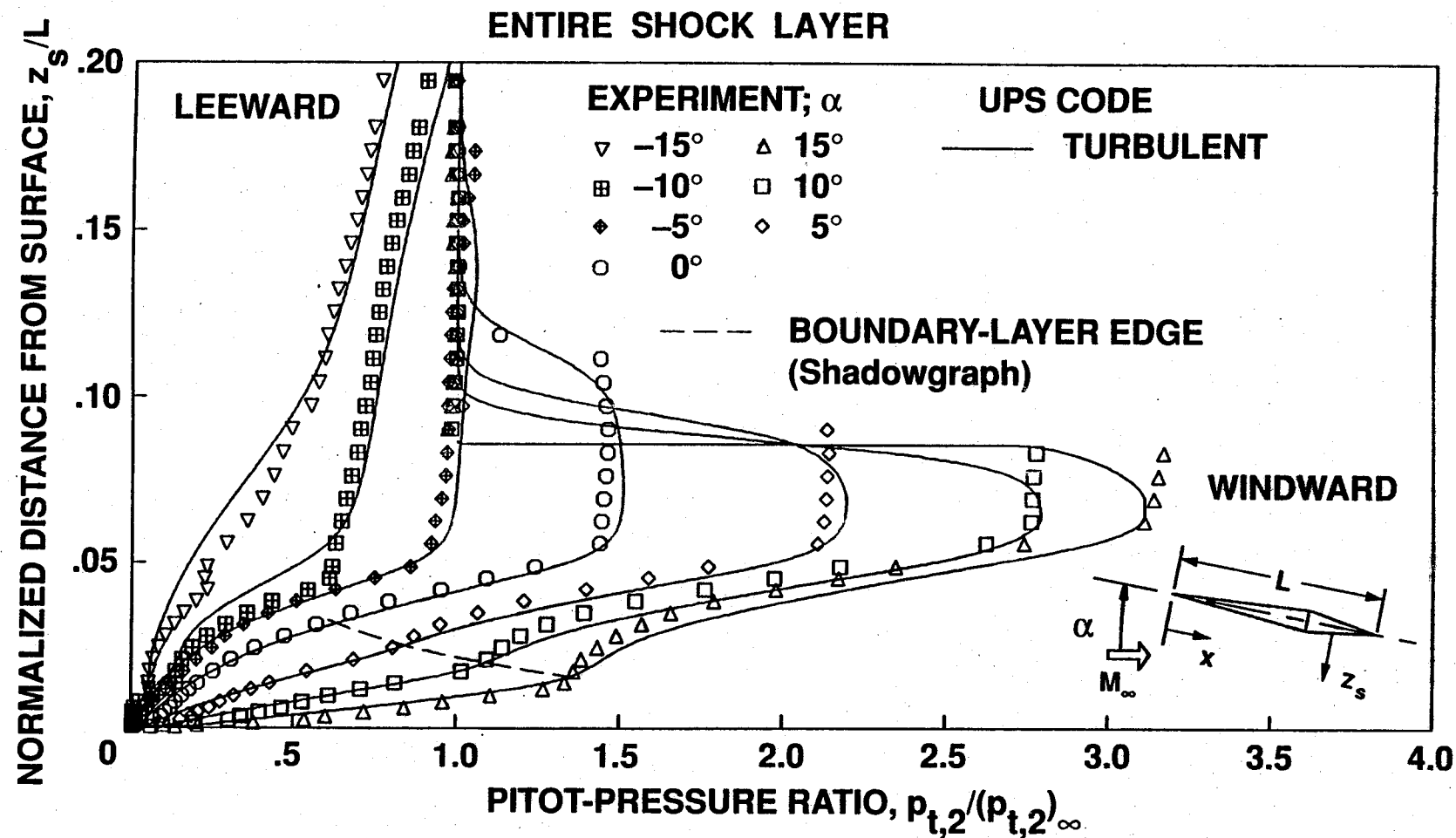


EXPERIMENTAL AND COMPUTATIONAL PITOT-PRESSURE DISTRIBUTIONS AT VARIOUS AFTERBODY STATIONS

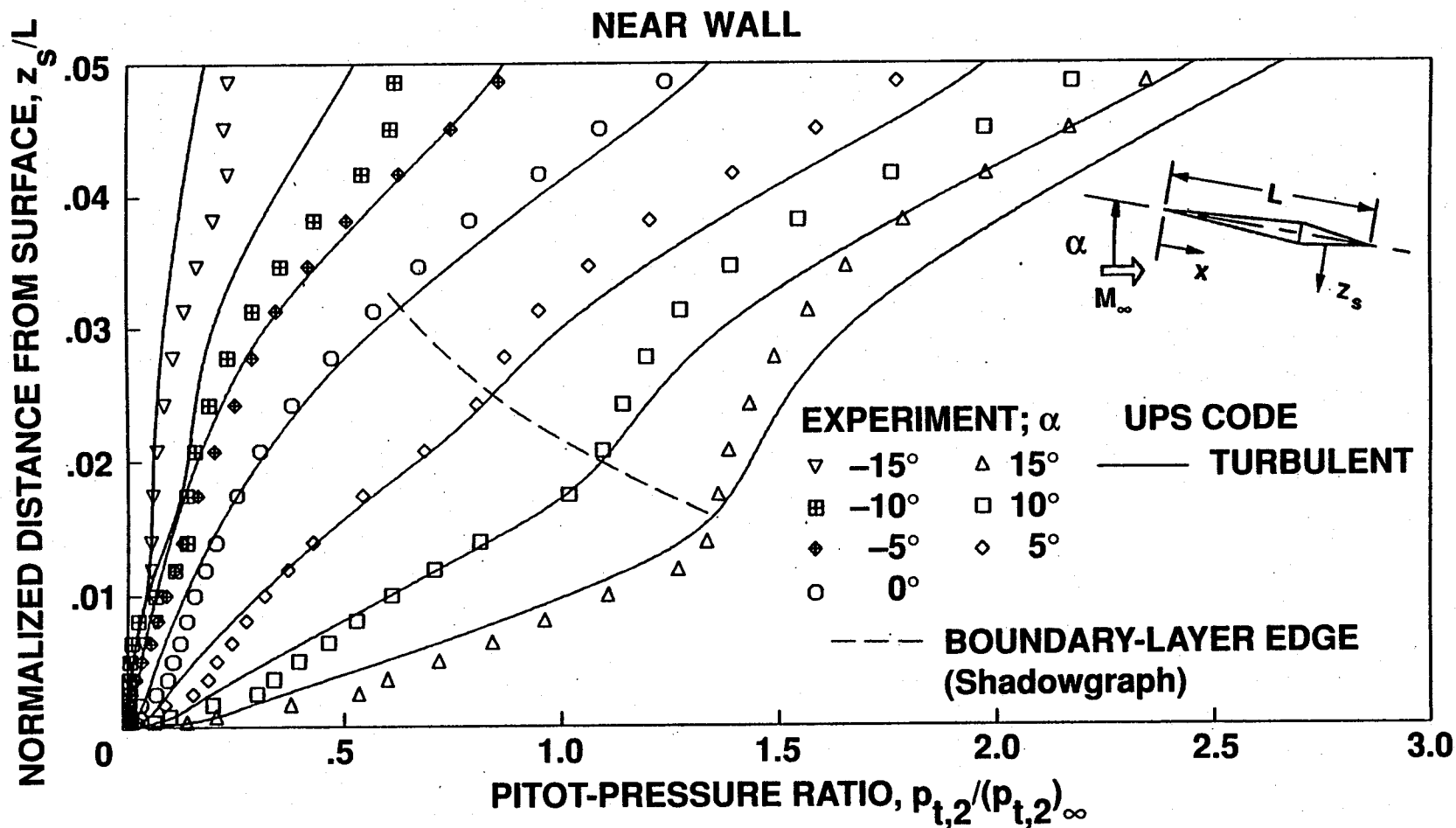
Centerline; $\alpha = 0^\circ$; $M_\infty = 7.4$; $Re_{\infty,L} = 15 \times 10^6$



EFFECT OF ANGLE OF ATTACK ON PITOT-PRESSURE DISTRIBUTIONS AT AFTERBODY STATION Centerline; $x/L = 0.8$; $M_\infty = 7.4$; $Re_{\infty,L} = 15 \times 10^6$



EFFECT OF ANGLE OF ATTACK ON PITOT-PRESSURE DISTRIBUTIONS AT AFTERBODY STATION Centerline; $x/L = 0.8$; $M_\infty = 7.4$; $Re_{\infty,L} = 15 \times 10^6$



CONCLUDING REMARKS

DESCRIBED EXPT. IN AMES 3.5-FT HWT WITH ALL-BODY MODEL

OUTLINED UPS CODE (Upwind PNS Solver)

PRESENTED EXPERIMENTAL & COMPUTATIONAL RESULTS:

- **Flow Visualization (Shadowgraphs & Oil-Flow Patterns)**
- **Surface Pressure Distributions**
- **Surface Convective Heat-Transfer Distributions**
- **Afterbody Pitot-Pressure Surveys**

OBSERVATIONS MADE:

- **Significant changes from forebody (conical) to afterbody (nonconical) flows**
- **Complex leeward flow at angle of attack with cross-flow separation and vortices**
- **Generally good agreement between experimental and UPS code results for:**
 - **Shock-Wave Angles**
 - **Surface Pressures (some differences at higher angles of attack and near leading edge)**
 - **Surface Heat Transfer (some differences for afterbody and leeward flows)**
 - **Afterbody Pitot-Pressure Surveys (some differences for leeward flow at higher angles of attack and for inner region of viscous layer)**

**COMPARISON OF EXPERIMENT WITH
CALCULATIONS
USING CURVATURE-CORRECTED TWO
EQUATION TURBULENCE MODELS FOR A TWO-
DIMENSIONAL U-DUCT**

**D.J. MONSON AND H.L. SEEGMILLER
NASA-AMES RESEARCH CENTER, MOFFETT FIELD, CA
P.K. MCCONNAUGHEY
NASA-MARSHALL SPACE FLIGHT CENTER, HUNTSVILLE, AL
Y.S. CHEN
SECA, INC., HUNTSVILLE, AL**

**2ND NASA CFD VALIDATION WORKSHOP
NASA LEWIS RESEARCH CENTER, JULY 10-12, 1990**



PREV. ANN.

AIAA 90-1484

**Comparison of Experiment with Calculations
Using Curvature-Corrected Zero and Two
Equation Turbulence Models for a Two-
Dimensional U-Duct**

D.J. Monson and H.L. Seegmiller

NASA-Ames Research Center, Moffett Field, CA

P.K. McConnaughey

NASA-Marshall Space Flight Center, Huntsville, AL

Y.S. Chen

SECA Inc., Huntsville, AL

**AIAA 21st Fluid Dynamics, Plasma Dynamics
and Lasers Conference**

June 18-20, 1990 / Seattle, WA

COMPARISON OF EXPERIMENT WITH CALCULATIONS USING CURVATURE-CORRECTED ZERO AND TWO EQUATION TURBULENCE MODELS FOR A TWO-DIMENSIONAL U-DUCT

D. J. Monson* and H. L. Seegmiller†

NASA Ames Research Center, Moffett Field, CA 94035

P. K. McConnaughey‡

NASA Marshall Space Flight Center, Huntsville, AL 35812

and

Y. S. Chen*

SECA Inc., Huntsville, AL 35805

Abstract

In this paper experimental measurements are compared with Navier-Stokes calculations using seven different turbulence models for the internal flow in a two-dimensional U-duct. The configuration is representative of many internal flows of engineering interest that experience strong curvature, such as that in the turnaround duct of the Space Shuttle Main Engine powerhead. A previous paper showed that application of a simple mixing length model to predict this flow gives poor agreement with important features of the experiment. In an effort to improve the agreement, this paper tests several versions of the two-equation $k - \epsilon$ turbulence model including the standard version, an extended version with a production range time scale, and a version that includes curvature time scales. Each are tested in their high and low Reynolds number formulations. Calculations using these new models and the original mixing length model are compared here with measurements of mean and turbulence velocities, static pressure and skin friction in the U-duct at two Reynolds numbers. The comparisons show that only the low Reynolds number version of the extended $k - \epsilon$ model does a reasonable job of predicting the important features of this flow at both Reynolds numbers tested.

Nomenclature

A^+	= 26
AR	= aspect ratio
C_f	= skin friction coeff. = τ_w/q_{ref}
C_p	= static pressure coeff. = $(p - p_{ref})/q_{ref}$
H	= channel height
k	= turbulent kinetic energy $\approx \frac{3}{4}(\langle u'^2 \rangle + \langle v'^2 \rangle)$
l	= mixing length

M	= Mach number
p	= static pressure
p_t	= total pressure
q	= dynamic pressure
R_c	= streamline radius of curvature
Re	= Reynolds number based on H and U_{ref}
r	= radial dist. from center of curvature
s	= down. dist. from channel entrance on duct c/l
U, V	= longitudinal, vertical mean velocities
u, v	= longitudinal, vertical inst. velocities
u', v'	= long., vert. inst. turb. vel. fluctuations
$u'v'$	= inst. turbulent Reynolds stress
u_τ	= wall friction velocity = $\sqrt{\tau_w/\rho}$
u^+	= dim. velocity = U/u_τ
x, y	= x, y coordinates
y_i	= distance from wall i in mixing length equation
y_i^+	= dim. wall variable for wall $i = y_i u_\tau / \nu$
δ	= boundary layer thickness
ϵ	= turbulent energy dissipation rate
θ	= angle into bend measured from bend entrance
κ	= von Karman's constant = 0.4
ν	= fluid kinematic viscosity
ν_t	= turbulent or eddy kinematic viscosity
ρ	= fluid density
τ_t	= turbulent shear stress = $-\rho \langle uv \rangle$
τ_w	= local wall shear stress or skin friction
ω	= mean flow vorticity
$\langle \rangle$	= RMS time average

Subscripts

i, o	= inner, outer walls
r, θ	= radial, tangential dir. in cyl. coord.
ref	= ref. conditions
s, n	= parallel, normal dir. in stream. coord.

* Research Scientist. Member AIAA

† Research Scientist

‡ Supervisory Aerospace Engineer. Member AIAA

I. Introduction

Computational fluid dynamics (CFD) is coming to play an increasingly major role in the initial design or verification of the internal flow in rocket engine components. For example, a three-dimensional incompressible Navier-Stokes code (INS3D) was recently used to guide a possible redesign of the hot gas manifold for the Space Shuttle Main Engine (SSME) powerhead. Changing from a three-circular-duct to a new two-elliptical-duct configuration greatly improved flow through the new powerhead as confirmed by experiment.^{1,2}

In spite of the above success, before CFD can be applied widely and with sufficient confidence for future rocket engine design, the codes must be calibrated and verified by comparing them against well-documented experimental data. Most engine components operate at very high pressures and flow Reynolds numbers (Re 's), so the flows are fully turbulent. Thus the accuracy of the codes is critically dependent on the turbulence models that they contain. Unfortunately, most turbulence models have been developed and verified only for flows with very mild extra strain rates. In contrast, the flows in rocket engine components may be subjected to very large extra strain rates or other perturbations. It is clear that knowledge on the structure of internal shear layers subject to such effects is crucial for successful numerical simulation of these flows. The accuracy of any turbulence model proposed for internal flow calculations should first be evaluated by comparing calculations using it with well-documented internal flow experiments containing one or more of the expected elements.

To date, few experimental studies of such internal flows have been reported. There are at least two reasons for this. First, CFD has only recently been applied to flows in rocket engines, so there has been little need for code verification experiments. Second, detailed measurements in rocket engines is usually quite difficult because of limited probe and/or optical access. One of the more widely-studied extra strain effects on internal flows is that arising from streamline curvature. There have been many such studies on the effects of mild curvature, and Bradshaw³ gives a comprehensive review of them. However, information on the effects of strong curvature and other large extra strain rates is very much needed.

Four recent studies have begun to examine both experimentally and theoretically some of the important effects present in strongly-curved internal flows. These studies investigate the internal flow in 180° turnaround or U-ducts (TAD's), where the radius ratio of bend centerline radius to duct height is of order unity. Such a geometry closely simulates many of the important features of the flow in the TAD of the SSME powerhead, such as the presence of strong curvature and pressure gradients, unsteady separation and interact-

ing shear layers on opposite walls.

In one study, Sharma et al.⁴ measured the flow using hot wires in an axisymmetric TAD air tunnel with $M = 0.1$ and $Re = 10^5$. (For reference, the flow in the SSME TAD is at $M = 0.1$ and $Re = 10^7$.) Chang and Kwak⁵ calculated this flow using INS3D with a simple mixing length turbulence model. Both experiment and theory indicated a small amount of separation on the inner (convex) wall near the end of the bend. Poor agreement was found for the velocities near the outer (concave) wall and for the turbulent shear stresses throughout the bend, however.

In a second study, Sandborn⁶ measured the flow using a one-component laser Doppler velocimeter (LDV) in a two-dimensional (2D) TAD water tunnel with $U_{ref} \approx 2$ m/s, and $Re = 7 \times 10^4$ to 5×10^5 . The results showed no inner wall separation at the lowest Re and greater separation than did Sharma et al.⁴ at the highest Re . Chen and Sandborn⁷ calculated this flow using a Navier-Stokes (NS) solver and the two-equation $k-\epsilon$ turbulence model both in its standard high Re form⁸, and also with a curvature correction.⁹ Few conclusions about the accuracy of the models could be drawn, however, since the calculations proceeded only to the 180° bend location. To that point the flow field is mainly pressure-driven and turbulence models are of less importance.

In a third study, Avva et al.¹⁰ calculated the TAD flow of Sandborn⁶ at a $Re = 9 \times 10^4$ using the standard $k-\epsilon$ model in both its high and low Re versions. At that Re , Sandborn's data showed only a small amount of separation on the inner wall at the bend exit. The low Re $k-\epsilon$ model predicted this, whereas the high Re model predicted no separation. In contrast, and to the puzzlement of the authors, the high Re model predicted the measured static pressure very well and the low Re one underpredicted it.

Finally, in a fourth study, Monson et al.¹¹ measured the flow using a two-component LDV in a 2D TAD air tunnel with $M = 0.1$ and $Re = 10^5$ and 10^6 . The measurements showed a small amount of inner wall separation at the lower Re , and a much larger amount at the higher Re . The flow was calculated using INS3D and a simple mixing length model. Poor agreement was found with the experiment in several respects. The static pressure drop in the bend was badly underpredicted, no separation was predicted at the higher Re , and the computed velocities near both walls were in error through most of the bend and further downstream. It appeared that the turbulence model was not reproducing the experiment very well, but no conclusions as to the cause of the failure could be drawn from the comparisons that were made.

Because of the generally poor agreement found by the above studies between theory and experiment for TAD-type flows, it is clear that improved turbulence models will be required to accurately compute them.

ther downstream are much too low. For the outer wall (Fig. 9b), the model indicates separation at the bend entrance, whereas none was measured. The model predicts very low C_f levels throughout the bend (as do all the models). Downstream C_f is also underestimated. The net result of the low C_f for the m.l. model is a prediction of too high a static pressure on the inner wall downstream of the bend (Fig. 10a). Note that the outer-wall pressure in the bend (Fig. 10b) is well-predicted by this and all of the other models. At $Re = 10^6$, the m.l. model underpredicts C_f on both walls (Fig.'s 9c, 9d) to an even greater extent than at the lower Re . Also, attached flow is indicated, whereas the experiment shows increased separation. The net result is, once again, too high a static pressure estimate downstream of the bend (Fig. 10c). It thus appears as though this zero-equation model, and others that employ the van Driest inner-scale mixing length (Eqn. 3), are not well-suited to calculating this TAD-type of flow.

Turning to the C_f results for the two standard $k-\epsilon$ models at $Re = 10^5$ in Fig. 9a, one can see that the low Re version (std. CH) shows early separation and later reattachment as compared to the experiment. (Note that the "waviness" of C_f predicted by the CH model and other ITW models was also seen by Patel et al.¹⁷ for b.l. flow in the presence of adverse pressure gradients.) A large overshoot in C_f then occurs. This is typical behavior of the standard Chien low Re model in reattachment regions.³⁰ The high Re model predicts separation length better, but undershoots measured C_f further downstream. For the outer wall (Fig. 9b), the CH model agrees quite well with measured downstream values, whereas the high Re model overpredicts the experiment. The net result for static pressure for these models (Fig. 10a), is that the CH version predicts too low a pressure downstream of the bend, and the WF version agrees with experiment. For $Re = 10^6$, both standard models indicate almost no separation (Fig. 9c), which is opposite the experimental Re trend. Downstream of the bend, the WF version predicts C_f well on the inner wall, and the ITW version agrees with experiment on the outer wall. The net result is that both versions predict inner-wall pressure (Fig. 10c) slightly higher than the measurements. It may be concluded that, although the standard $k-\epsilon$ models predict the TAD flow better than the m.l. model, neither version consistently predicts measured C_f , C_p and extent of separation at both Re tested. Recent calculations (not included in this paper) of this TAD flow using the "LB1"¹⁷ low Re model rather than the CH model in std. $k-\epsilon$ show much improved C_f and C_p predictions. The prediction of separation extent is not improved, however.

The next models to be considered for their C_f and C_p behavior are the two versions of the curvature-corrected $k-\epsilon$ model. Recall that the goal of these

models is to decrease TKE and TSS on the convex wall to be more consistent with experimental observations. It is obvious from the C_f data in Fig.'s 9a and 9c that the models grossly overcorrect for such effects. Much too large an extent of separation is predicted at both Re . Following reattachment, both models also suffer from the same problems as did the standard models (i.e., an overshoot of C_f for the ITW version and an undershoot for the WF version). The net result is a prediction of too much downstream pressure loss for both versions at both Re (Fig.'s 10a and 10c). These results indicate that curvature corrections developed for flows with small curvature don't always apply to flows with large curvature like the present TAD-type of flow. Thus, the curvature $k-\epsilon$ model of Park and Chung²⁵ cannot be recommended for this flow.

Finally, consider the skin friction and pressure predictions of the extended $k-\epsilon$ models. At $Re = 10^5$, the ITW version indicates slightly more separation than the experiment (Fig. 9a), but predicts the downstream recovery of C_f very well. The WF version shows much larger separation and an undershoot of downstream C_f . For the outer wall (Fig. 9b), the ITW model once again predicts downstream C_f and the WF model shows high values. For C_p (Fig. 10a), the ITW model predicts pressure exactly and the WF model shows too much pressure loss. For $Re = 10^6$, the ITW model predicts extent of separation and downstream C_f almost exactly (Fig. 9c). (This is the only model tested that doesn't show decreased separation at this Re .) The WF version once again shows very slow reattachment and downstream C_f recovery. Both versions of this model show excellent agreement with measured pressure at this Re (Fig. 10d). The conclusion can be reached that the extended $k-\epsilon$ model in its low Re version is the only one tested in this study that predicts most of the important measured features of this TAD-type of flow at both Re . The high Re version using the wall function formulation of Ref. 20 does not do as well, however. Perhaps a more sophisticated wall function treatment of the separated region which includes a wake-like parameter³⁰ in the "law-of-the-wall", and/or better choice of near-wall grid spacing in certain regions of the TAD flow, would provide improved results. Overall, the extended $k-\epsilon$ model together with the "LB1" low Re model overcomes the two main shortcomings of the standard CH $k-\epsilon$ model for this TAD flow: 1) It reduces TKE and TSS on the convex wall so that separation is predicted at both Re ; and 2) It eliminates the large C_f overshoot downstream of reattachment. Even though it is not developed specifically for curved flows, the extended LB $k-\epsilon$ model seems to capture most of the important features of this flow and yet retains complete generality. Other turbulence models that reduce the level of turbulent stresses near the convex wall in the bend could perhaps do equally well when combined with the "LB1" low Re model.

V. Summary and Conclusions

A joint experimental and computational study of the flow in a two-dimensional U-duct with very strong curvature has been carried out. The significant conclusions of this study are as follows:

1. The experiment shows: a) significant turbulence enhancement on the outer (concave) wall consistent with a previously-observed curvature instability mechanism; b) almost total destruction of turbulence on the inner (convex) wall consistent with previous studies of flows with much less curvature; c) separation on the inner wall at the bend exit that increases with Reynolds number; and d) extreme core turbulence downstream of the bend that creates linear "plug-flow" type velocity profiles.
2. The mixing length model of Patankar et al.²², and the standard and curvature-corrected $k - \epsilon$ models used with the low Reynolds number model of Chien¹⁸, failed to predict this flow and the trends with Reynolds number.
3. The extended $k - \epsilon$ turbulence model of Chen and Kim²⁴, combined with the low Reynolds number model of Lam and Bremhorst¹⁹, was the only turbulence model of those evaluated that predicted measured extent of separation, skin friction and static pressure throughout all regions of the flow at both low and high Reynolds number. The good prediction of skin friction downstream of the bend on the concave wall side may mean that modeling the details of that complex flow in the bend itself may not be required.
4. Application of wall functions as formulated by Launder and Spalding²⁰ to the above extended model produced poor results. More sophisticated treatment of the separated region and better choice of near-wall grid spacing in some regions of the flow may be required to improve predictions using wall functions.

References

1. Chang, J. L. C., Kwak, D., Dao, S. C. and Rosen, R., "A Three-Dimensional Incompressible Flow Simulation Method and its Application to the Space Shuttle Main Engine, Part I-Laminar Flow," AIAA Paper 85-0175, Reno, Nevada, Jan. 1985.
2. Chang, J. L. C., Kwak, D., Dao, S. C. and Rosen, R., "A Three-Dimensional Incompressible Flow Simulation Method and its Application to the Space Shuttle Main Engine, Part II-Turbulent Flow," AIAA Paper 85-1670, Cincinnati, Ohio, July 1985.
3. Bradshaw, P., "Effect of Streamline Curvature on Turbulent Flow," AGARD-AG-169, August 1973.
4. Sharma, L., Ostermier, B., Nguyen, L., Dang, P. and O'Conner, G., "Turbulence Measurements in an Axisymmetric Turnaround Duct Air Flow Model," Rocketdyne Div., Rockwell Inter., Report RSS-8763, ATU-87-5237, Oct. 1987.
5. Chang, J. L. C. and Kwak, D., "A Numerical Study of Turbulent Internal Shear Layer Flow in an Axisymmetric U-Duct," AIAA Paper 88-0596, Reno, Nevada, Jan. 1988.
6. Sandborn, V. A., "Measurement of Turbulent Flow Quantities in a Rectangular Duct with a 180 Degree Bend," NASA Conf. Publ. 3012, in *Advanced Earth-to-Orbit Propulsion Technology 1988*, Vol. II, Proc. of a Conf. at NASA Marshall Space Flight Center, May 1988, pp. 292-304.
7. Chen, Y. S. and Sandborn, V. A., "Computational and Experimental Study of Turbulent Flows in 180 Degree Bends," AIAA Paper 86-1516, Huntsville, Ala., June 1986.
8. Jones, W. P. and Launder, B. E., "The Prediction of Laminarization with a Two-Equation Model of Turbulence," *Int. Jour. of Heat and Mass Transfer*, Vol. 15, No. 2, 1972, pp. 301-314.
9. Rodi, W., "A New Algebraic Relation for Calculating the Reynolds Stresses," *ZAMM*, Vol. 56, 1976, pp. T219-T221.
10. Avva, R., Smith, C. and Singhal, A., "Comparative Study of High and Low Reynolds Number Versions of $k - \epsilon$ Models," AIAA Paper 90-0246, Reno, Nevada, Jan. 1990.
11. Monson, D. J., Seegmiller, H. L. and McConnaughey, P. K., "Comparison of LDV Measurements and Navier-Stokes Solutions in a Two-Dimensional 180-Degree Turn-Around Duct," AIAA Paper 89-0275, Reno, Nevada, Jan. 1989.
12. Monson, D. J., "A Nonintrusive Laser Interferometer Method for Measurement of Skin Friction," *Experiments in Fluids*, Vol. I, 1983, pp. 15-22.
13. Chen, Y. S., "Compressible and Incompressible Flow Computations with a Pressure Based Method," AIAA Paper 89-0286, Reno, Nevada, Jan. 1989.
14. Issa, R. I., "Solution of the Implicit Discretised Fluid Flow Equations by Operator-Splitting," *Jour. Comp. Phys.*, Vol. 62, 1985, pp. 40-65.
15. Stone, H. L., "Iterative Solution of Implicit Approximations of Multidimensional Partial Differential Equations," *SIAM Jour. Numer. Anal.*, Vol. 5, 1968, pp. 530-556.
16. Peric, M., "Efficient Semi-Implicit Solving Algorithm for Nine-Diagonal Coefficient Matrix," *Numerical Heat Transfer*, Vol. 11, 1987, pp. 251-279.
17. Patel, V. C., Rodi, W. and Scheuerer, G., "Turbulence Models for Near-Wall and Low Reynolds Number Flows: A Review," *AIAA Jour.*, Vol. 23, No. 9, Sept. 1985, pp. 1308-1319.
18. Chien, K. Y., "Predictions of Channel and Boundary-Layer Flows with a Low-Reynolds-Number Turbulence Model," *AIAA Jour.*, Vol. 20, No. 1, Jan. 1982, pp. 33-38.

19. Lam, C. K. G. and Bremhorst, K., "A Modified Form of the $k - \epsilon$ Model for Predicting Wall Turbulence," *Jour. Fluids Engrg., Trans. of ASME*, Vol. 103, Sept. 1981, pp. 456-460.
20. Launder, B. E. and Spalding, D. B., "The Numerical Computation of Turbulent Flows," *Comp. Meth. Appl. Mech. and Engrg.*, Vol. 3, 1974, p. 269-289.
21. Williams, R., ED32, NASA-MSFC, Private Communication, June 1990.
22. Patankar, S. V., Ivanovic, M. and Sparrow, E. M., "Analysis of Turbulent Flow and Heat Transfer in Internally Finned Tubes and Annuli," *Jour. Heat Transfer*, Vol. 101, Feb. 1979, pp. 29-37.
23. Burke, R. W., "Computation of Turbulent Incompressible Wing-Body Junction Flow," AIAA Paper 89-0279, Reno, Nevada, Jan. 1989.
24. Chen, Y. S. and Kim, S. W., "Computation of Turbulent Flows Using an Extended $k - \epsilon$ Turbulence Closure Model," NASA CR-179204, Oct. 1987.
25. Park, S. W. and Chung, M. K., "Curvature-Dependent Two-Equation Model for Prediction of Turbulent Recirculating Flows," *AIAA Jour.*, Vol. 27, No. 3, March 1989, pp. 340-344.
26. Launder, B. E., Pridden, C. H. and Sharma, B. I., "The Calculation of Turbulent Boundary Layers on Spinning and Curved Surfaces," *Jour. Fluids Engrg., Trans. of ASME*, Vol. 99, March 1977, pp. 231-239.
27. Barlow, R. S. and Johnston, J. P., "Roll-Cell Structure in a Concave Turbulent Boundary Layer," AIAA Paper 85-0297, Reno, Nevada, Jan. 1985.
28. Gillis, J. C., Johnston, J. P., Moffat, R. J. and Kays, W. M., "Experimental Data and Model for the Turbulent Boundary Layer on a Convex, Curved Surface," NASA CR-3391, March 1981.
29. Schlichting, H., *Boundary Layer Theory*, Fourth Edition, McGraw-Hill Book Co., Inc., 1960, p.303.
30. Viegas, J. R., Rubesin, M. W. and Horstman, C. C., "On the Use of Wall Functions as Boundary Conditions for Two-Dimensional Separated Compressible Flows," AIAA Paper 85-0180, Reno, Nevada, Jan. 1985.

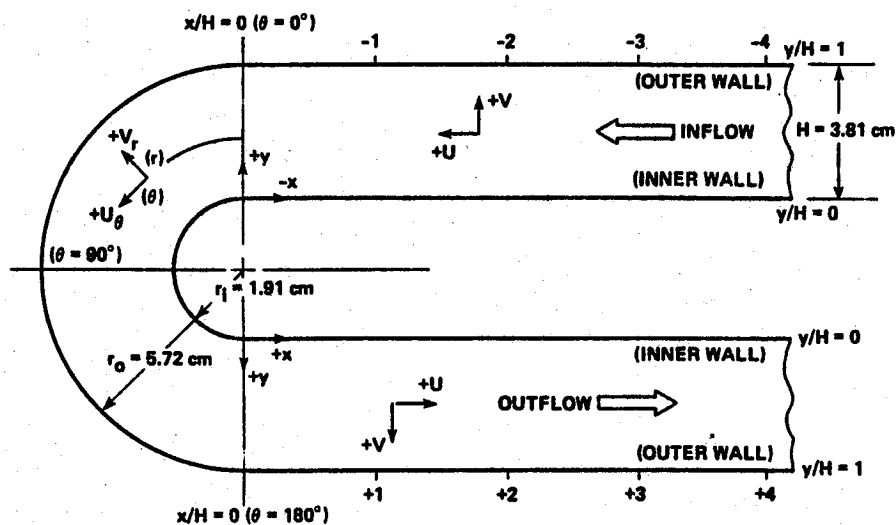


Fig. 1 Coordinate system for Ames HRC I turnaround duct.

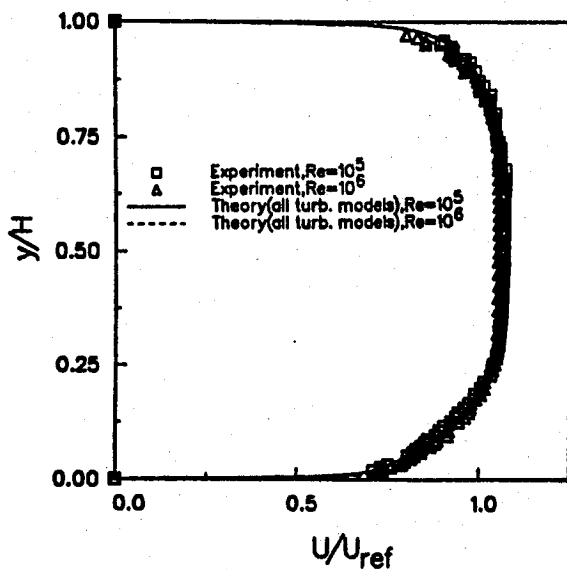


Fig. 2a Inlet longitudinal velocity in turnaround duct, $M_{ref} = 0.1$, $z/H = -4$.

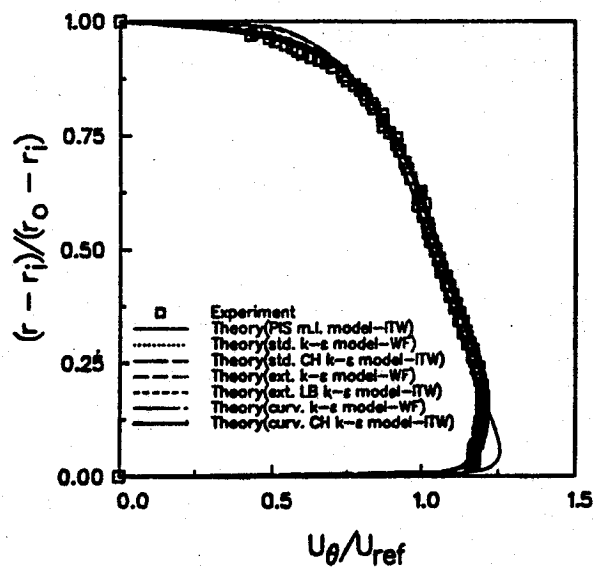


Fig. 3a Longitudinal velocity in turnaround duct, $Re = 10^6$, $\theta = 0^\circ$.

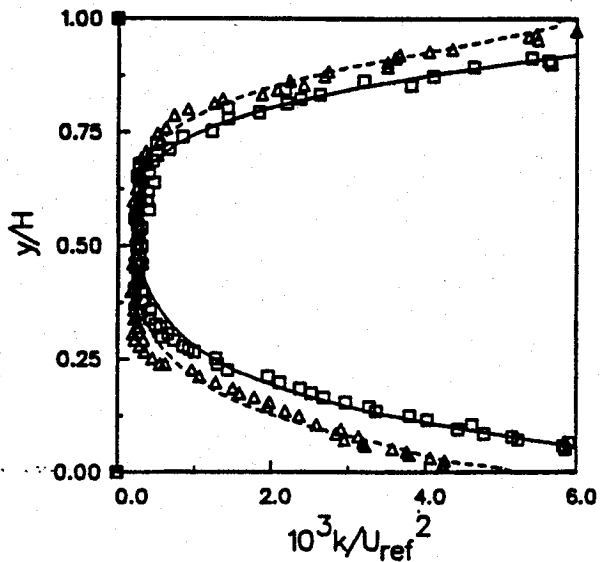


Fig. 2b Inlet turbulent kinetic energy.

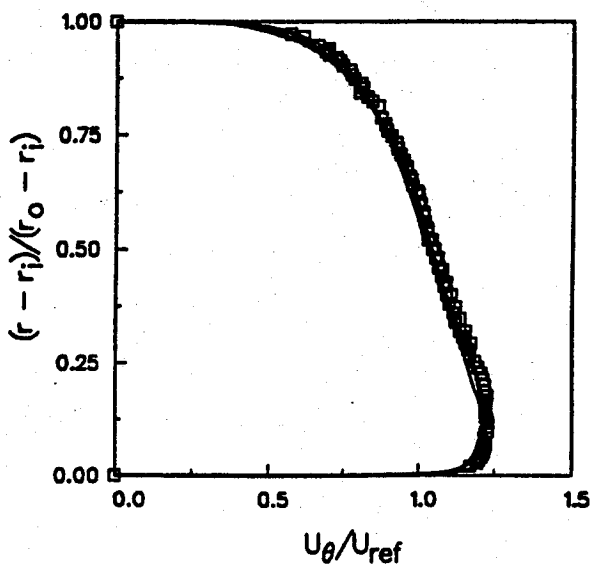


Fig. 3b $Re = 10^6$.

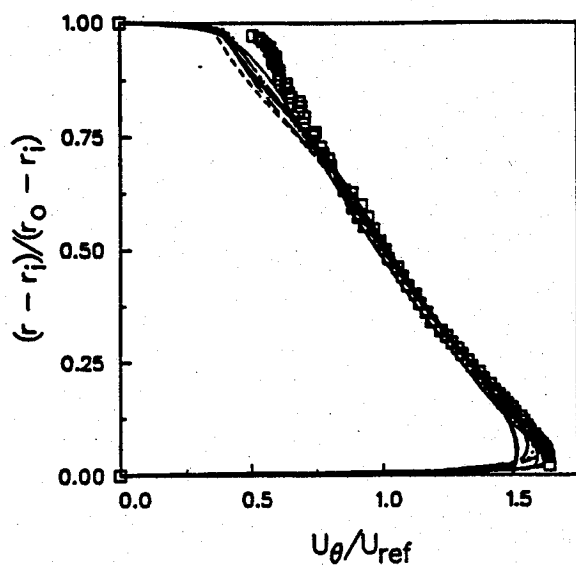


Fig. 4a Longitudinal velocity in turnaround duct, $Re = 10^5$, $\theta = 90^\circ$, (legend as in Fig. 3a).

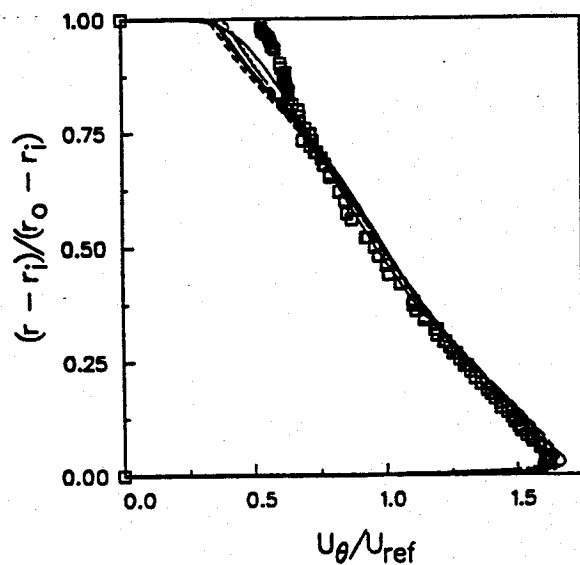


Fig. 4b $Re = 10^6$.

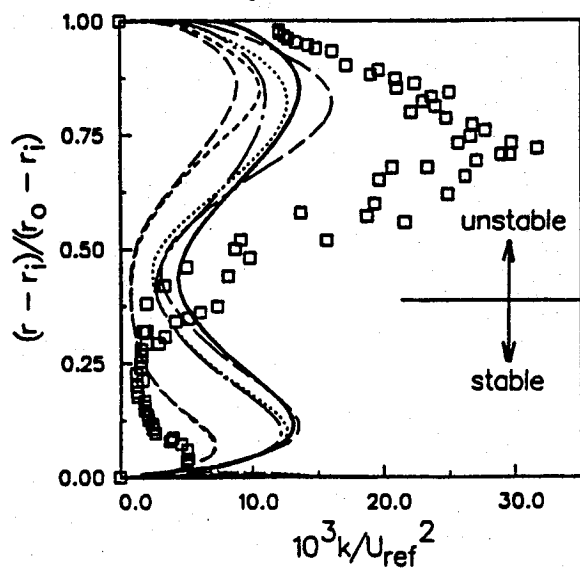


Fig. 4c Turbulent kinetic energy.

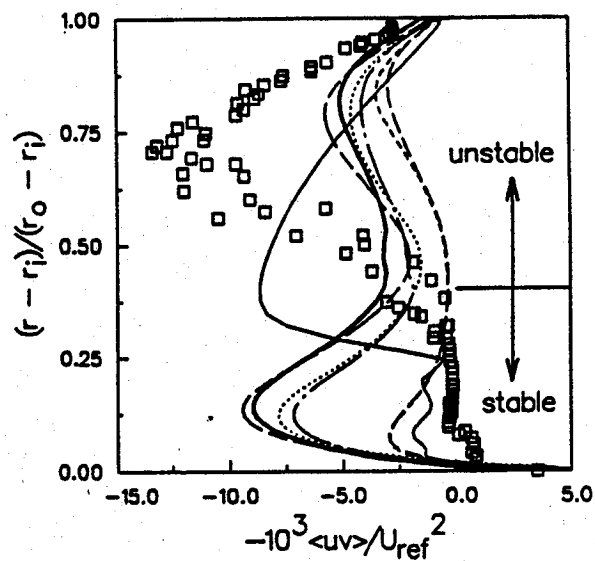


Fig. 4d Turbulent shear stress.

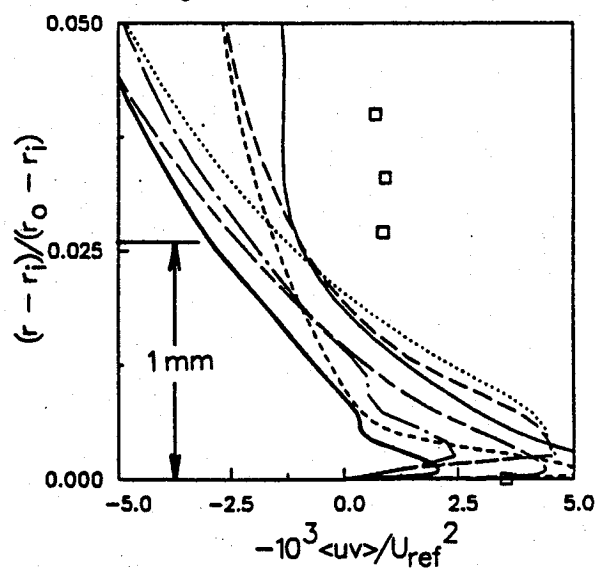


Fig. 4e Turbulent shear stress near convex wall.

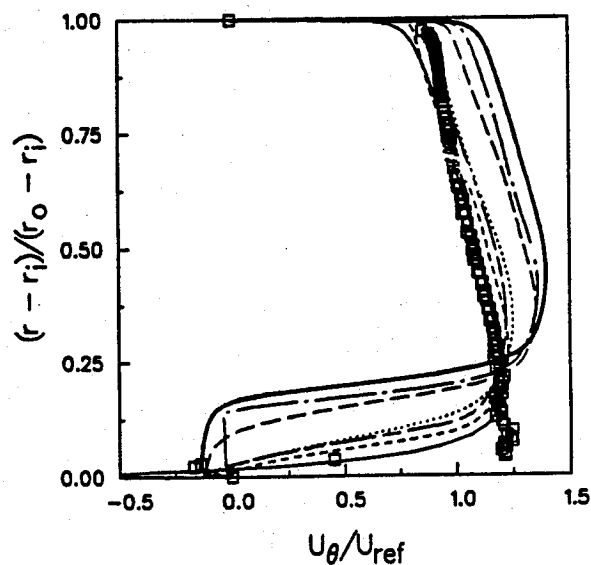


Fig. 5a Longitudinal velocity in turnaround duct, $Re = 10^5$, $\theta = 180^\circ$, (legend as in Fig. 3a).

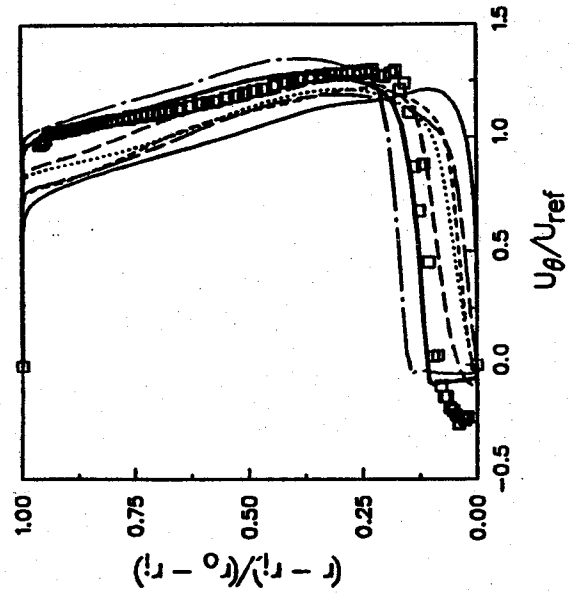


Fig. 5b $Re = 10^6$.

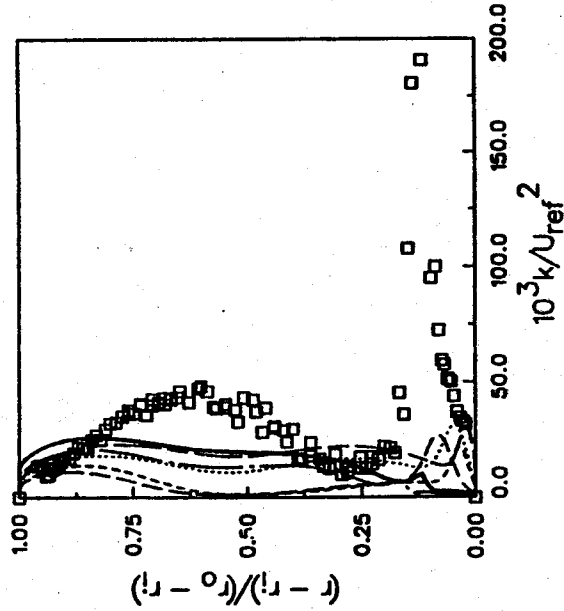


Fig. 5c Turbulent kinetic energy.

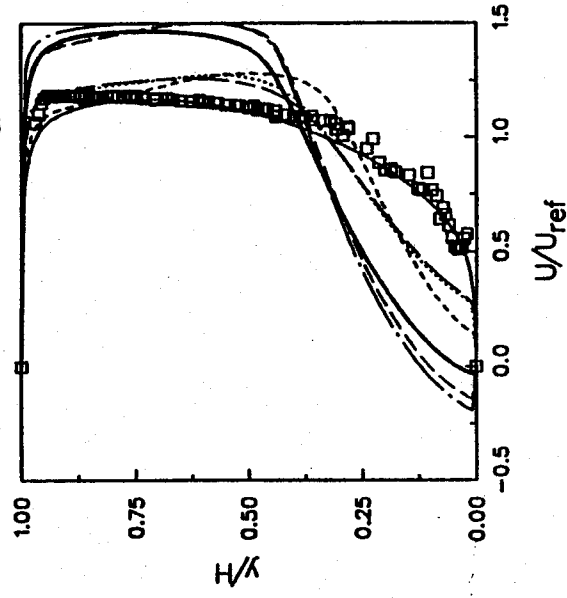


Fig. 6a Longitudinal velocity in turnaround duct, $Re = 10^5$, $z/H = 2$, (legend as in Fig. 3a).

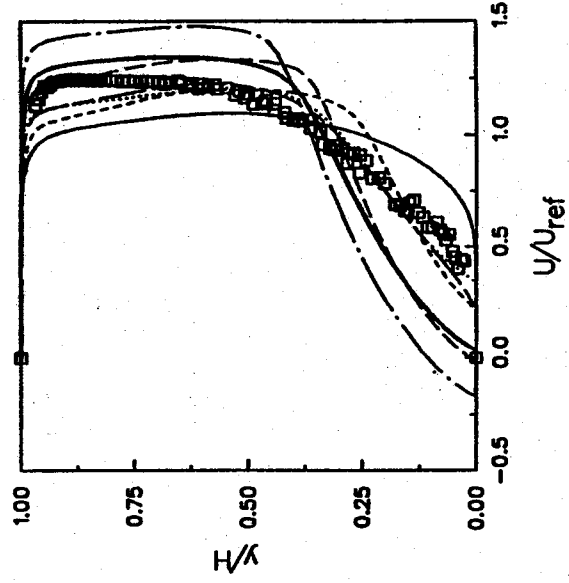


Fig. 6b $Re = 10^6$.

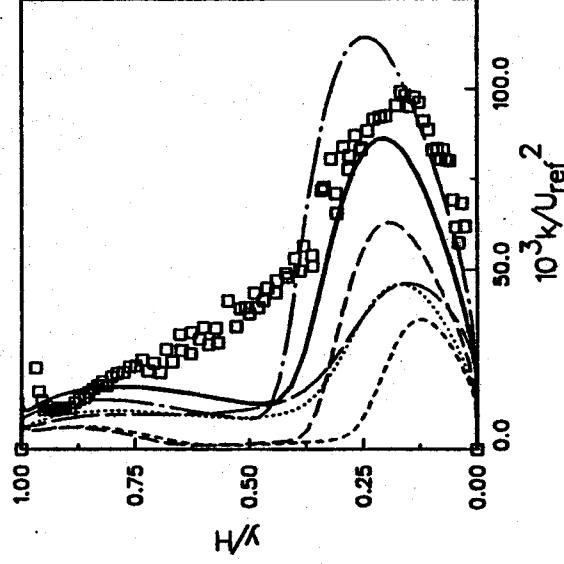


Fig. 6c Turbulent kinetic energy.

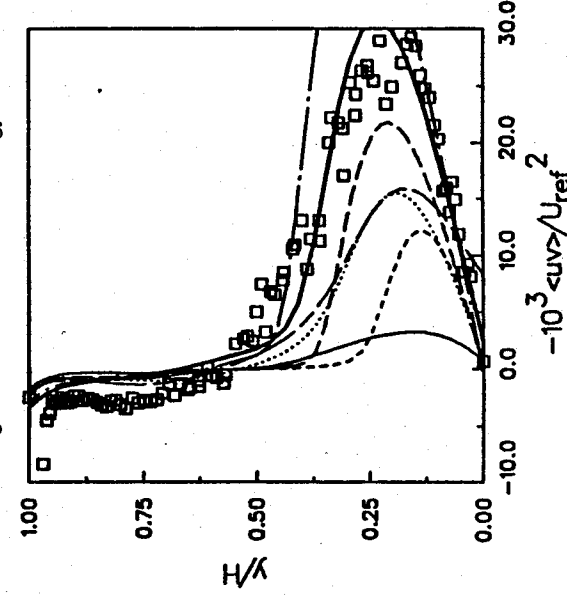


Fig. 6d Turbulent shear stress.

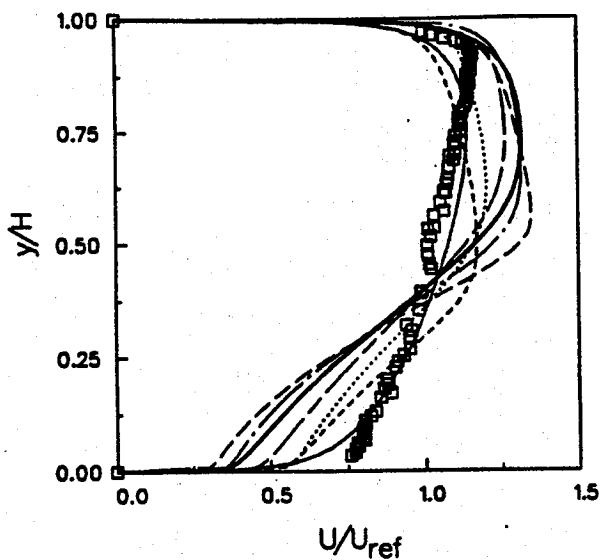


Fig. 7a Longitudinal velocity in turnaround duct, $Re = 10^5$, $z/H = 4$, (legend as in Fig. 3a).

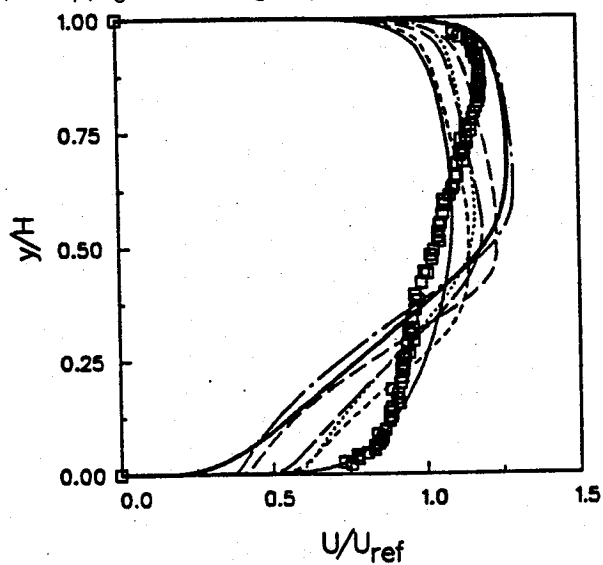


Fig. 7b $Re = 10^6$.

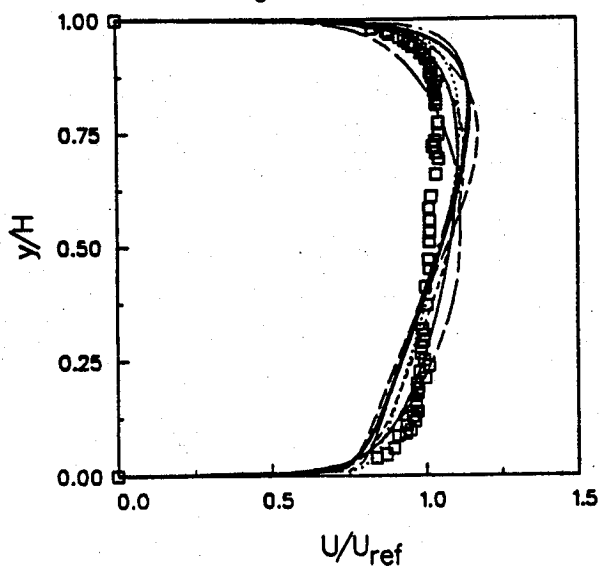


Fig. 8a Longitudinal velocity in turnaround duct, $Re = 10^5$, $z/H = 12$, (legend as in Fig. 3a).

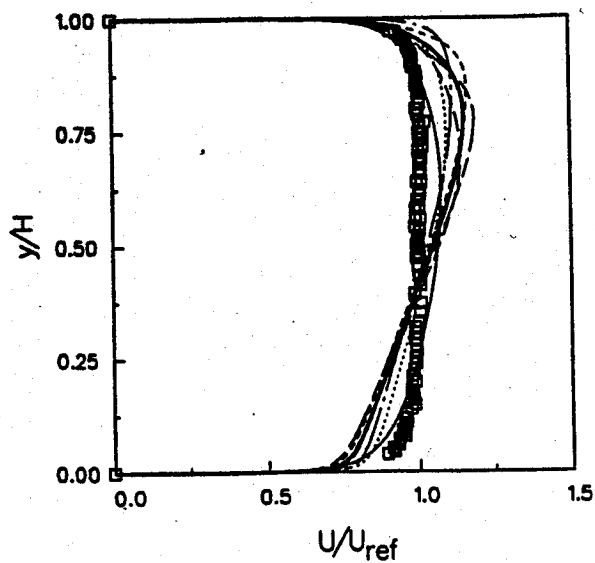


Fig. 8b $Re = 10^6$.

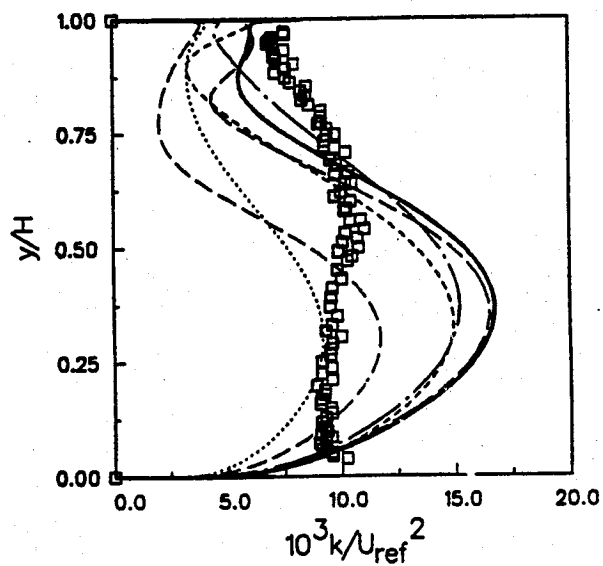


Fig. 8c Turbulent kinetic energy.

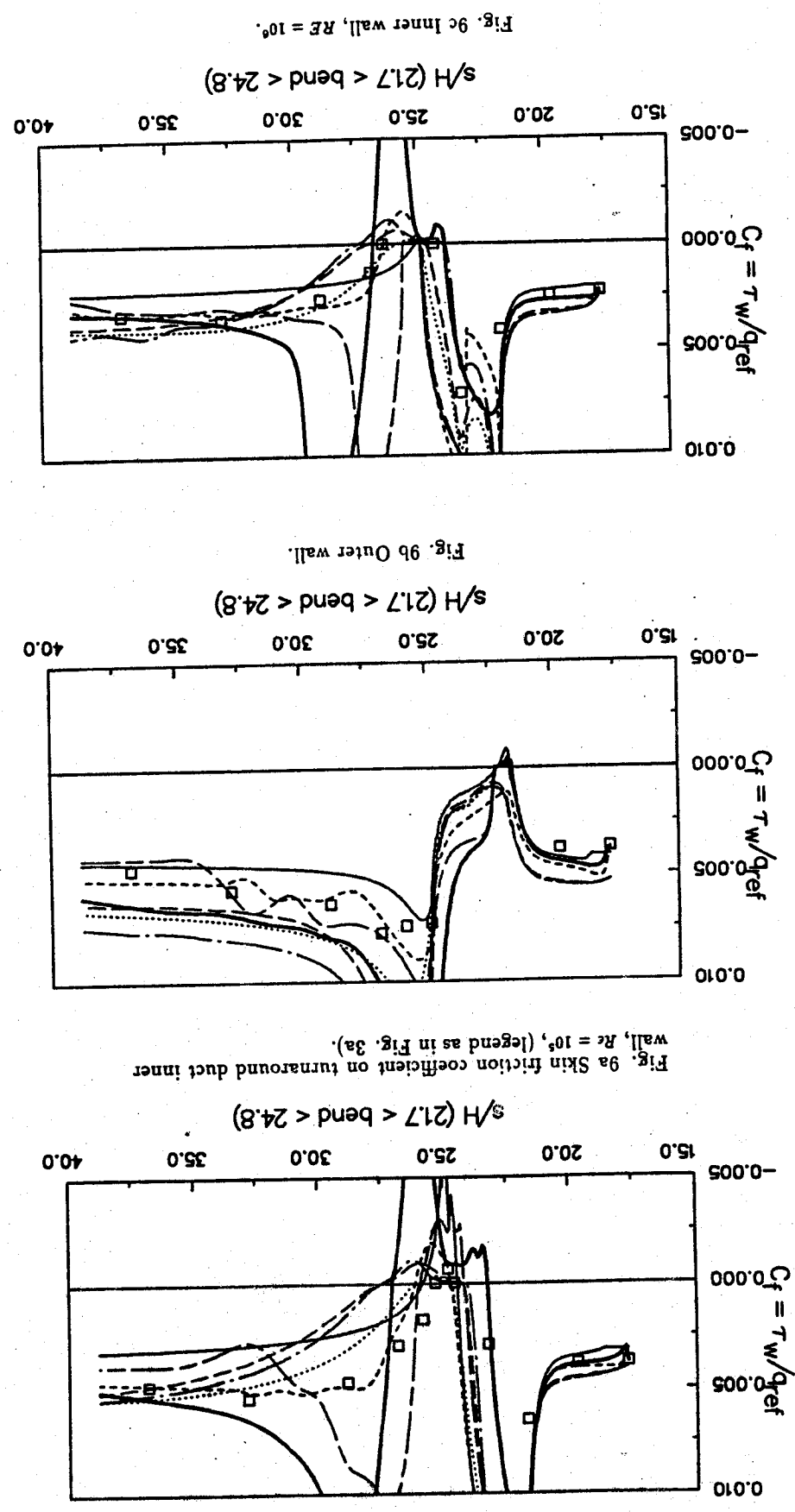


Fig. 9a Skin friction coefficient on turnaround duct inner wall, $Re = 10^5$, (legend as in Fig. 3a).

Fig. 9b Outer wall.

Fig. 9c Inner wall, $Re = 10^5$.

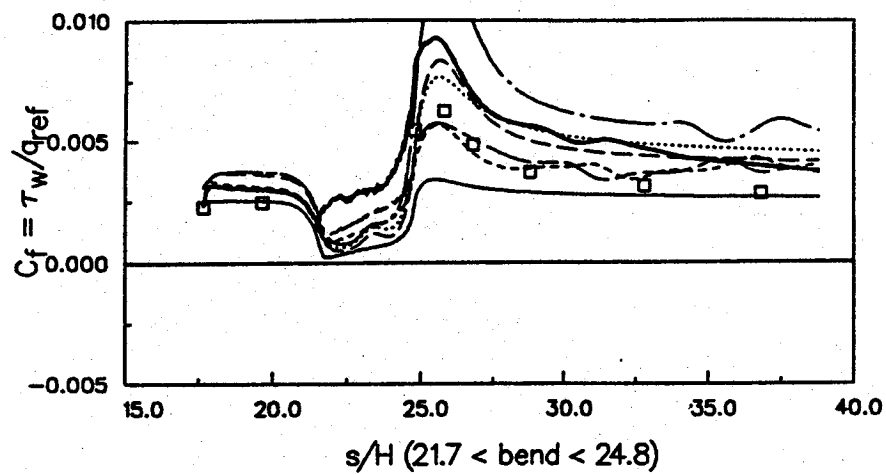


Fig. 9d Outer wall.

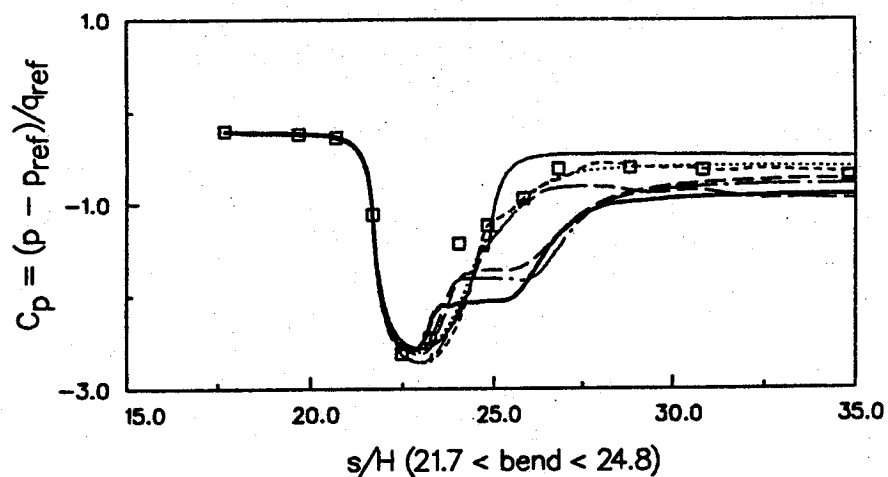


Fig. 10a Static pressure coefficient on turnaround duct inner wall, $Re = 10^5$, (legend as in Fig. 3a).

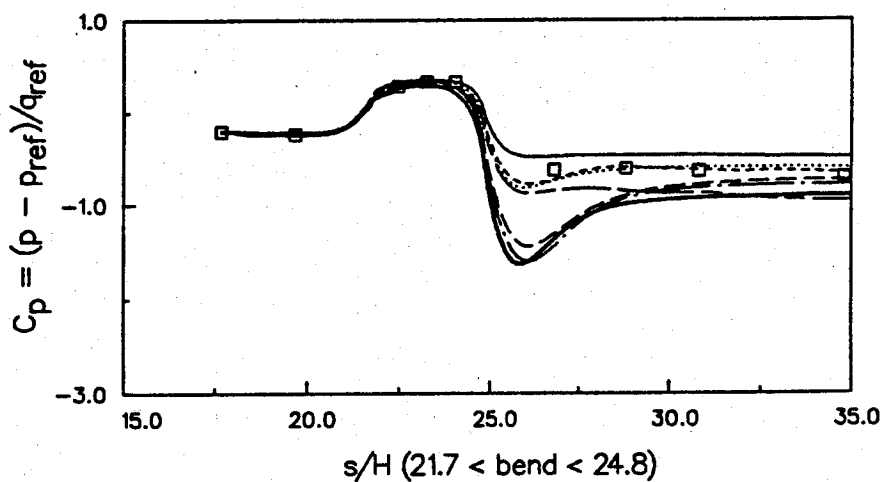


Fig. 10b Outer wall.

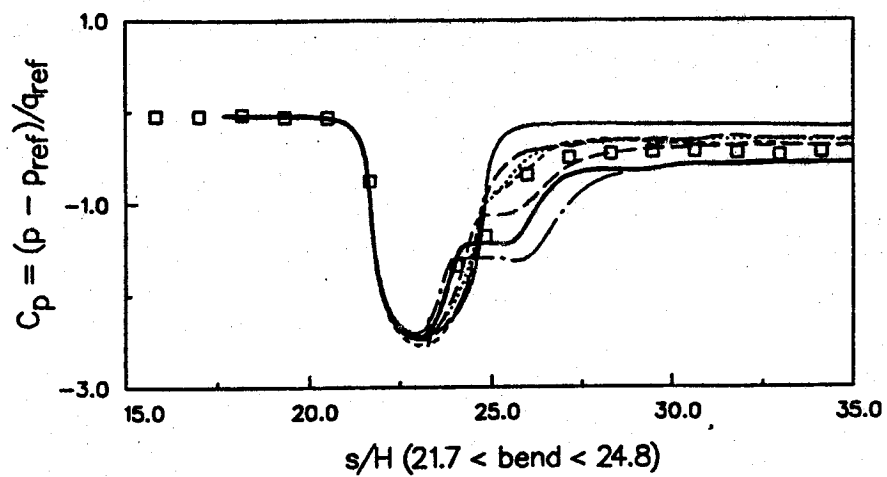


Fig. 10c Inner wall, $Re = 10^6$.

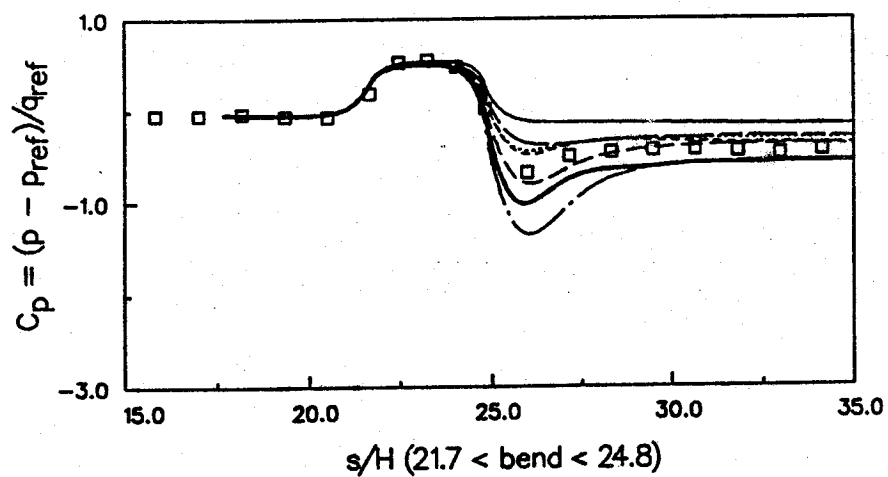


Fig. 10d Outer wall.

56-34
116676 P-16

CFD Valid Workshop

Flow over a Rearward Facing Step

Scott O. Kjelgaard

Experimental Methods Branch

Fluid Mechanics Division

Second NASA CFD Validation Workshop

NASA - Lewis Research Center

July 10-12, 1990

Outline

- **Objective/Approach**
- **Experimental Set-up**
- **Global Flowfield Measurements**
- **Detailed Profiles**
- **Concluding Remarks**

Langley Investigation

Instrumentation Techniques used:

- 3-component Laser Velocimeter
- 2-component hot wire
- Pitot pressure surveys
- 4th LV component for measuring freestream velocity

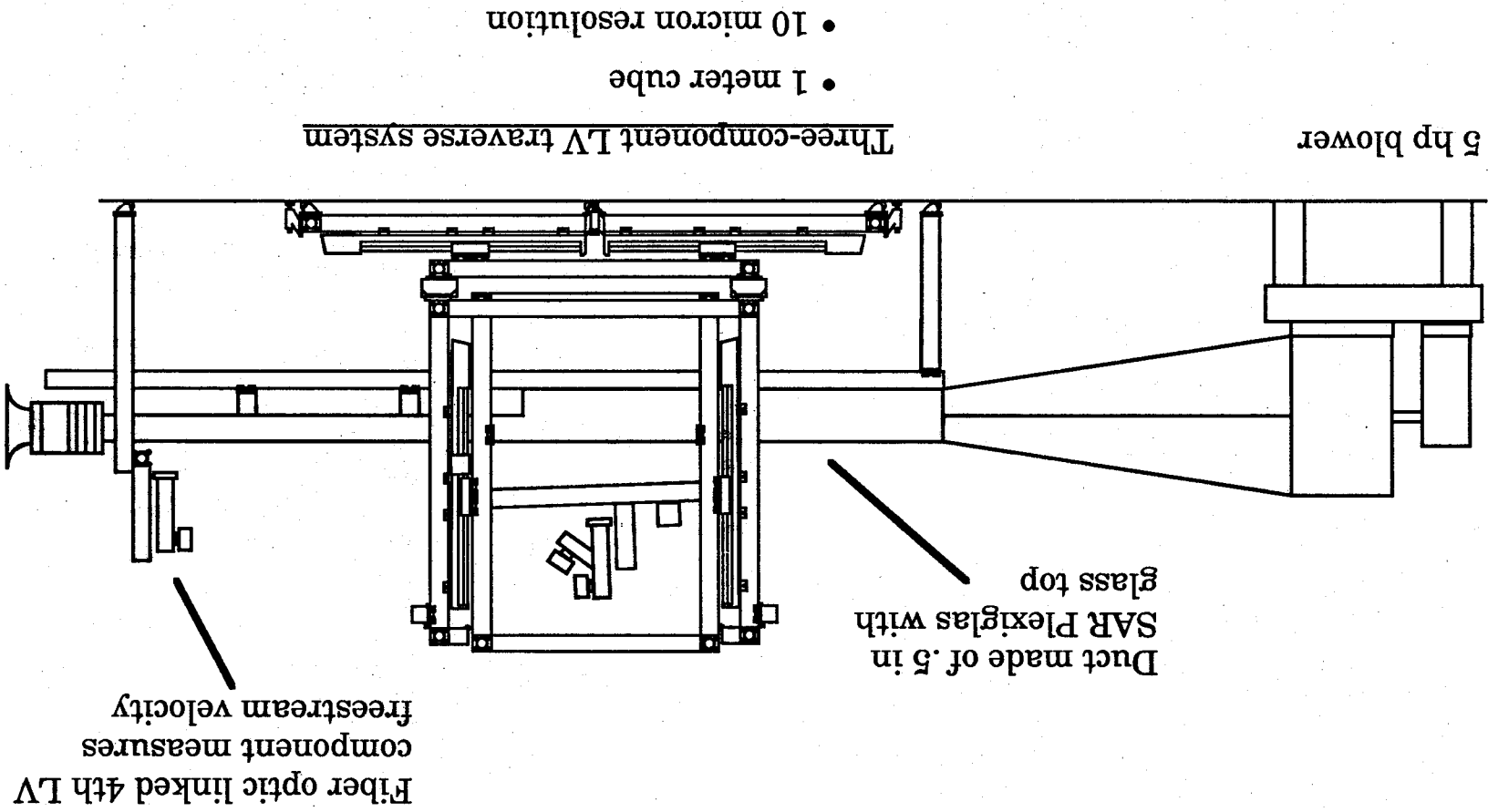
Measurements to be made:

- | | |
|--------------------------|--------------|
| • Mean Velocity | Pitot, LV |
| • Fluctuating Velocities | LV, Hot wire |
| • Spectral Information | LV, Hot wire |

Measurement locations:

- Global surveys - 700 pt surveys on grid (0.5 inch spacing)
- Detailed surveys at $x/H = -3, -2, -1, \dots, 8, 9$ (0.06 inch spacing)
- Spectral data at $x/H = 0, 7$

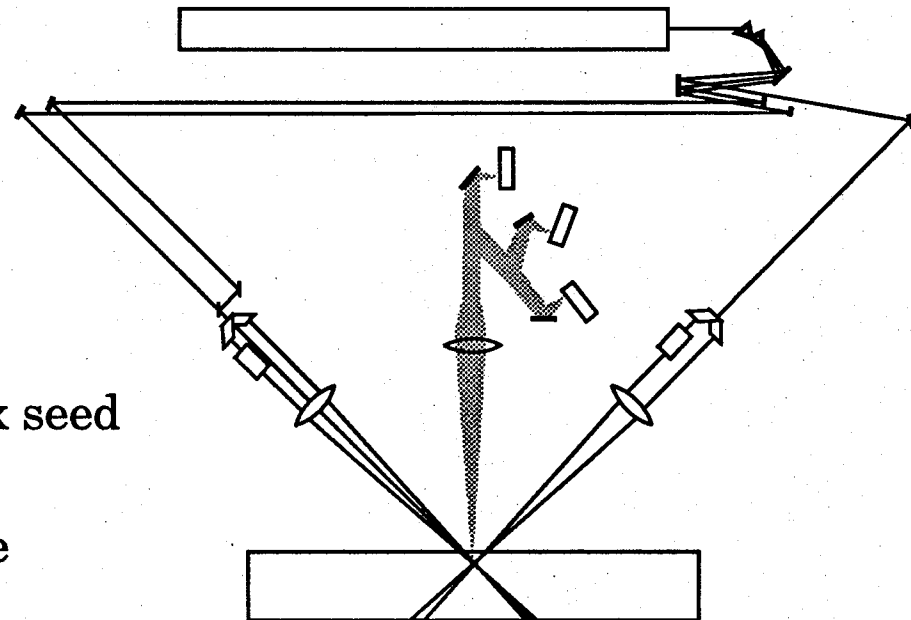
Schematic Of Rearward Facing Step Experiment



LV System Description

Three-component Laser Velocimeter

- 5 watt Argon ion laser
- color separation (514.5, 496.5, 476.5 nm wavelengths)
- sample volume size ≈ 100 microns
- 0.8 micron polystyrene latex seed particle
- data acquired in coincidence
- orthogonal transmit optics
- receive optics backscatter at 45 degrees



Fourth-component Laser Velocimeter

- fiber-optic link using 488 nm wavelength

681

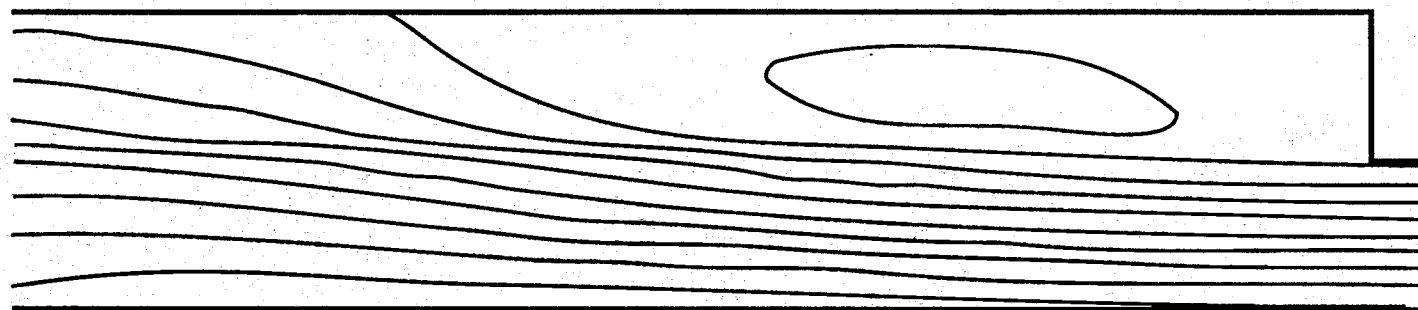
July 1990

7/9/90

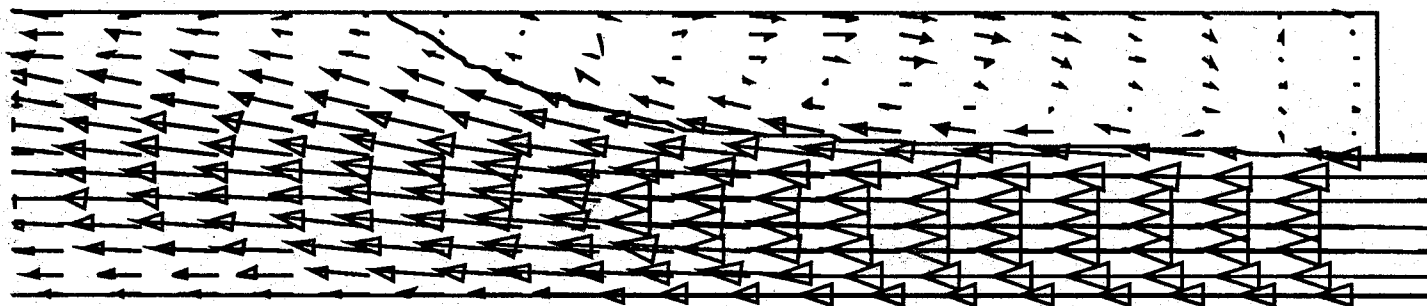
8

H/x

6 8 7 6 5 4 3 2 1 0



Streamlines



Cross-flow Velocity Vectors

Mean Flowfield Characteristics

CFD Valid Workshop

Rearward Facing Step Investigation

U COLOR CONTOUR PLOT



-0.23

1.00

V COLOR CONTOUR PLOT



-0.13

0.10

W COLOR CONTOUR PLOT



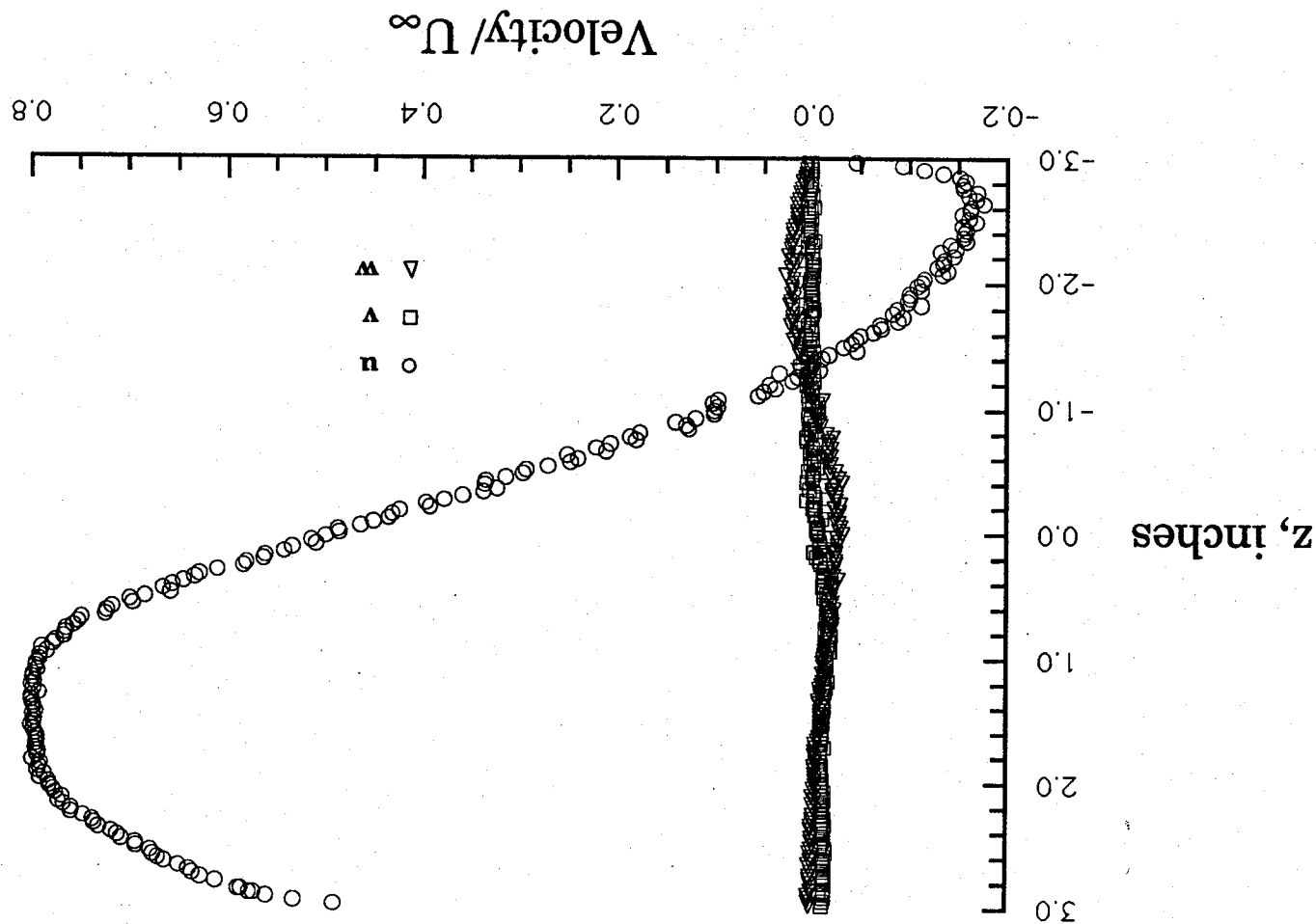
-0.11

0.06

191

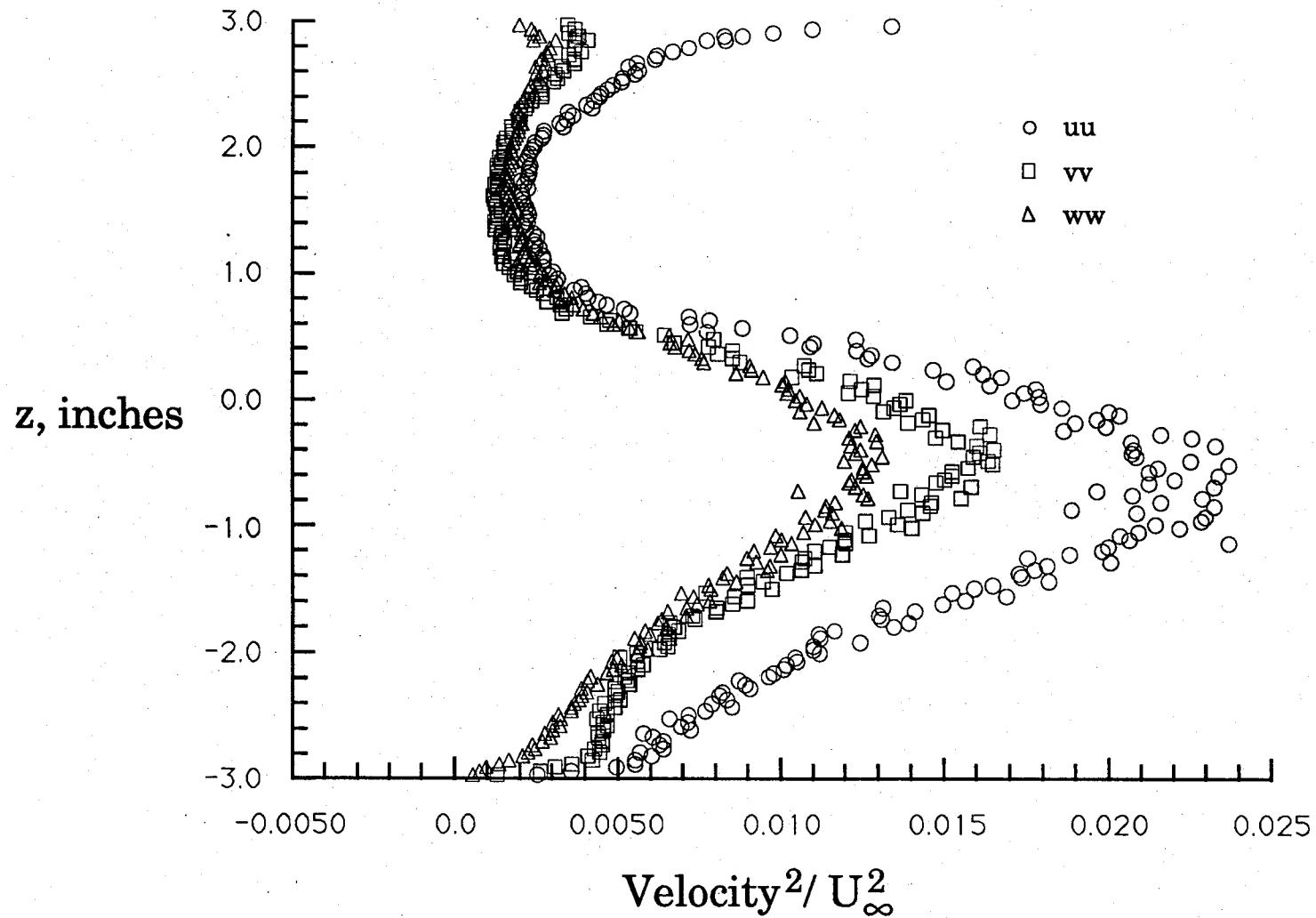
July 1990

Mean Velocity Profile at $x/H = 3$



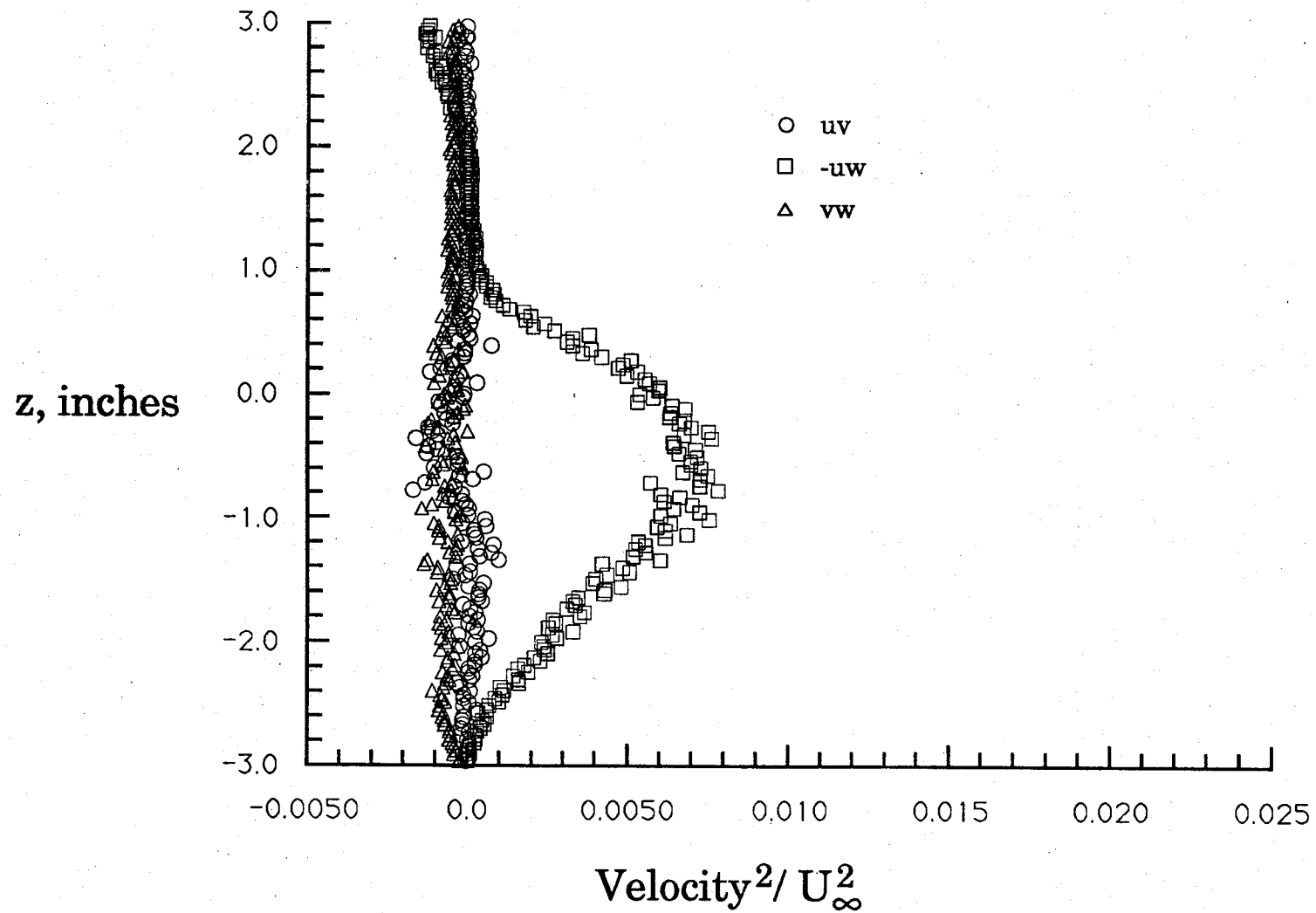
261

Normal Stress Profile at $x/H = 3$

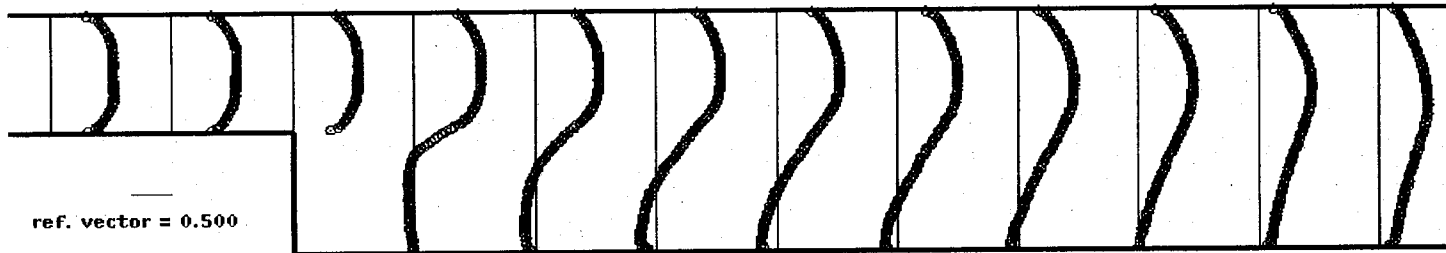


Σ 61

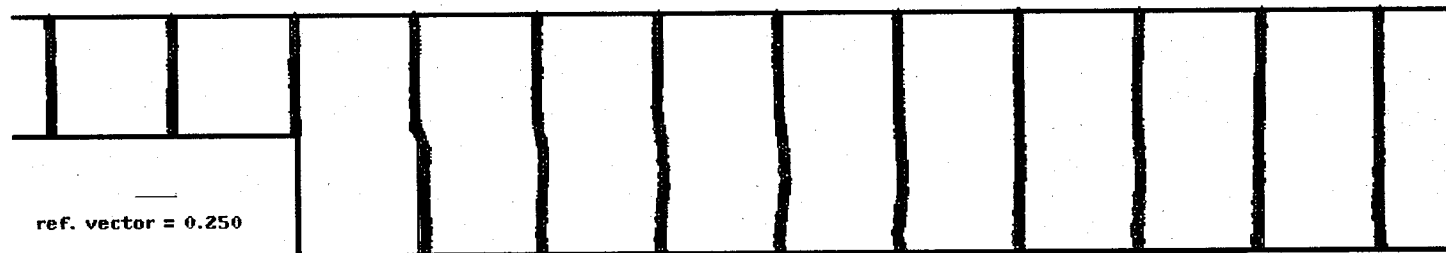
Shear Stress Profile at $x/H = 3$



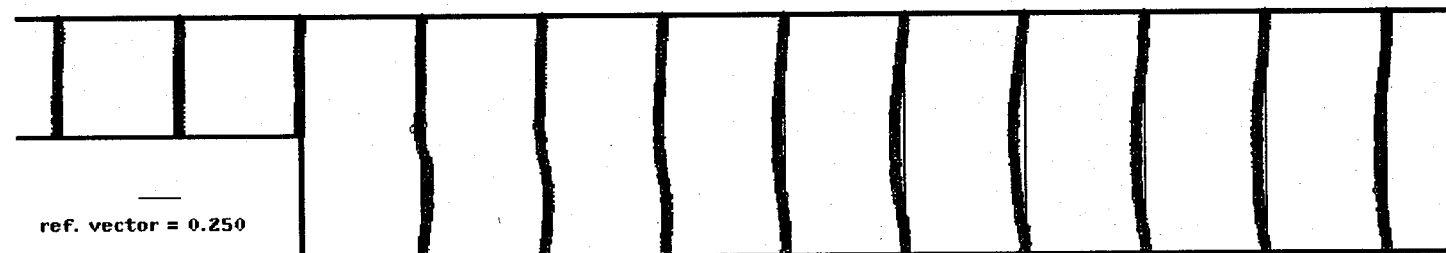
Rearward Facing Step Investigation



Mean u

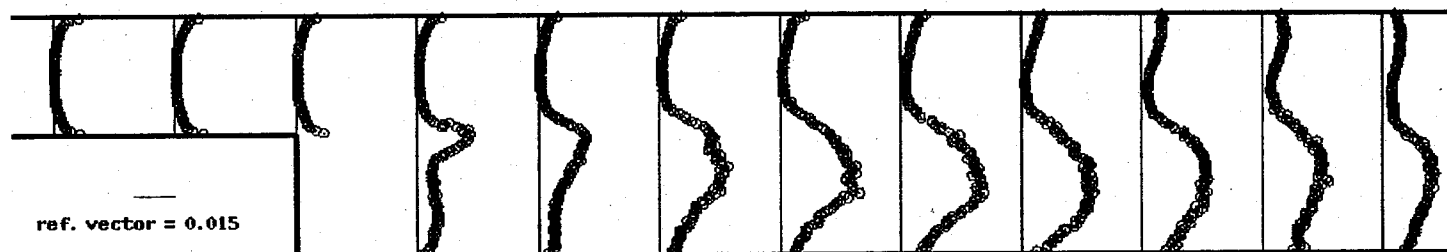


Mean v

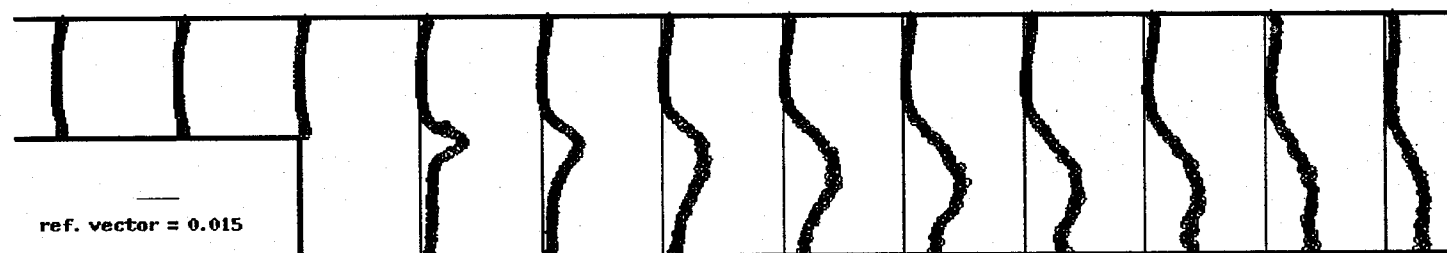


Mean w

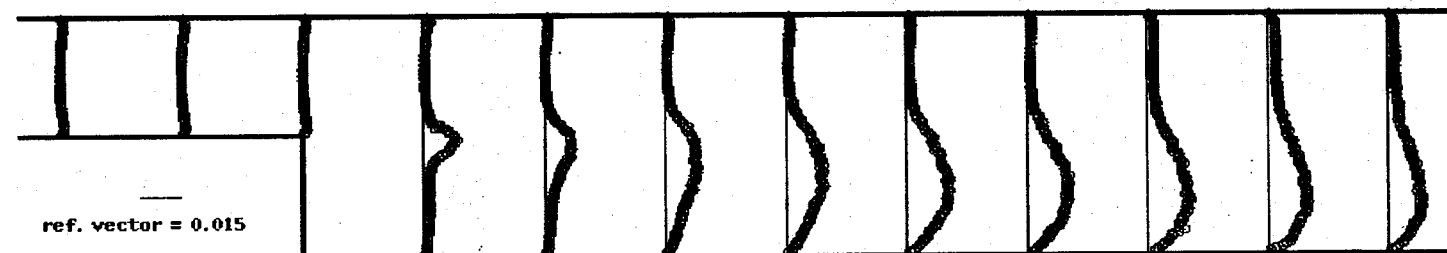
Rearward Facing Step Investigation



Mean uu

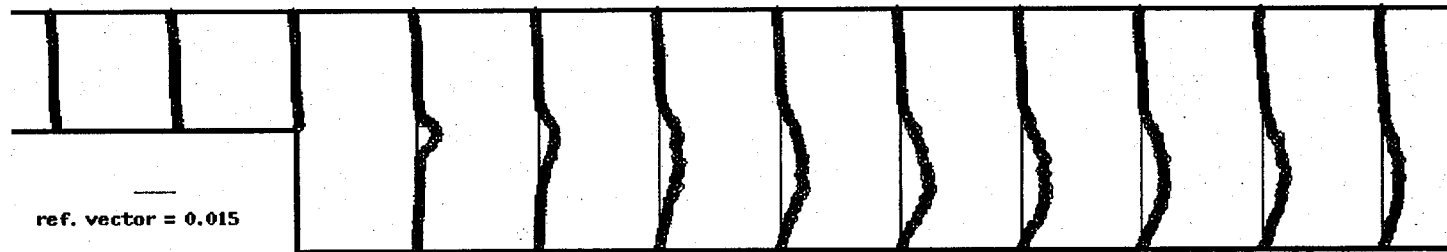


Mean vv

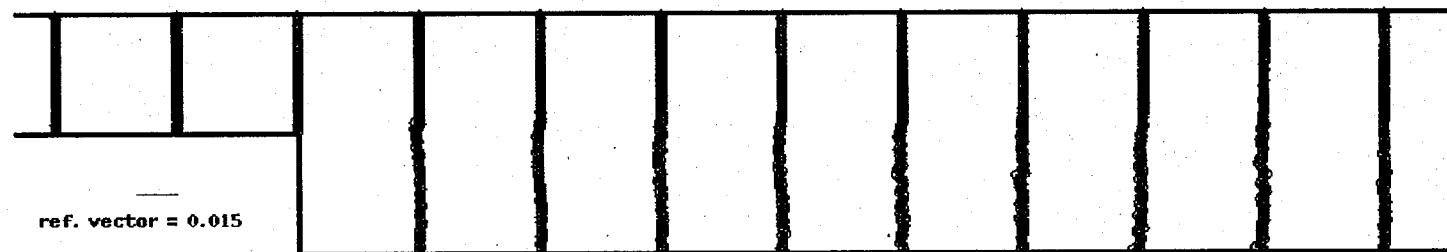


Mean ww

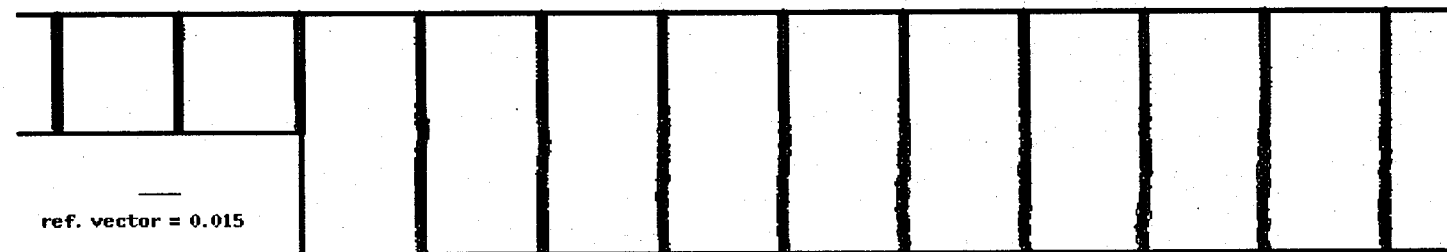
Rearward Facing Step Investigation



-Mean uw



Mean uv



Mean vw

Concluding Remarks

An investigation of the flow over a rearward facing step is currently underway.

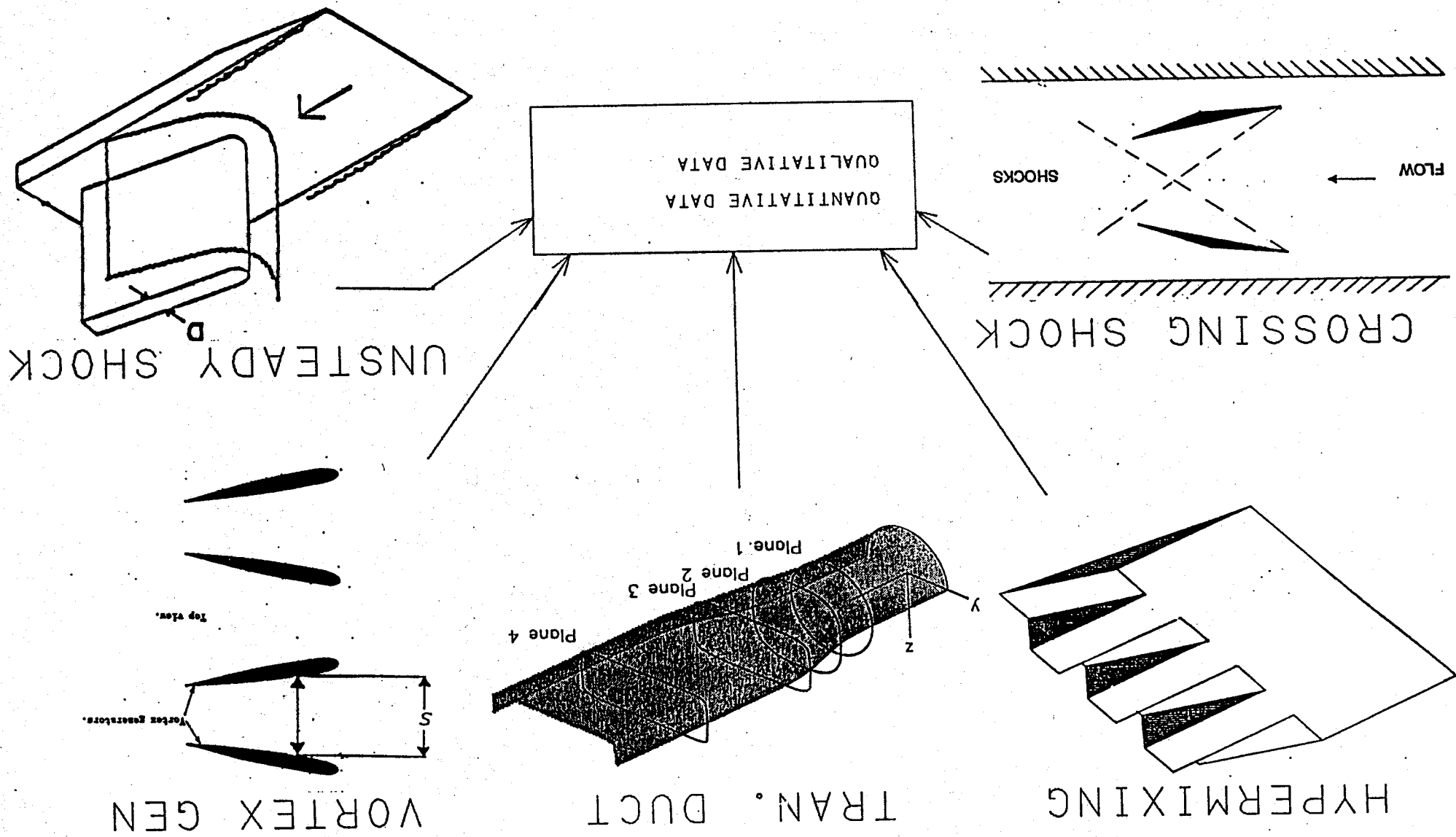
- A facility was constructed to meet the requirements of the DNS
- Three-component LV data has been acquired throughout the flow documenting
 - Mean Velocities
 - Full Reynolds stress tensor
- Preliminary review of the data by CFD personnel has expanded the scope to include additional Reynolds numbers and ER
- Next phase of the test should begin Jan. 91

LeRC/IFMD INLET DUCT AND NOZZLE HIGH SPEED VALIDATION EXPERIMENTS

- VALIDATION WORKSHOP EXPERIMENTS
 - CROSSING SHOCKS/BOUNDARY-LAYER INTERACTION
 - LeRC/UNIVERSITY UNSTEADY SHOCK/BOUNDARY-LAYER INTERACTION
 - VORTEX GENERATORS
- ADDITIONAL VALIDATION EXPERIMENTS
 - HIGH SPEED MIXING
 - TRANSITION DUCTS

N93-70582

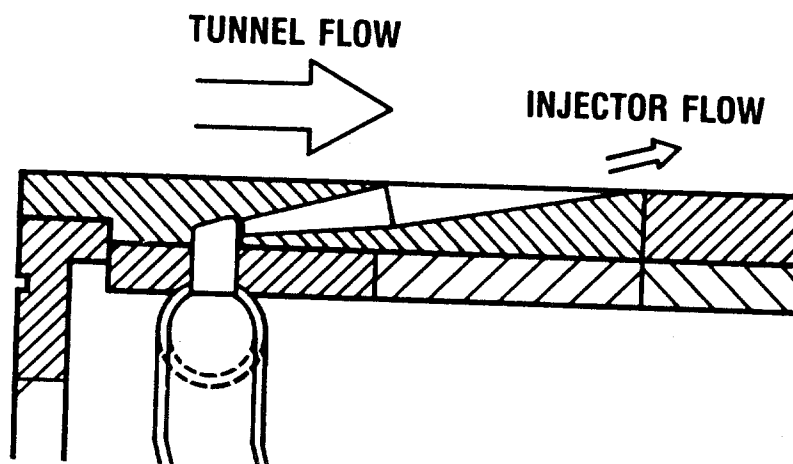
EXPERIMENTS IN 3-D FLUID MECHANICS



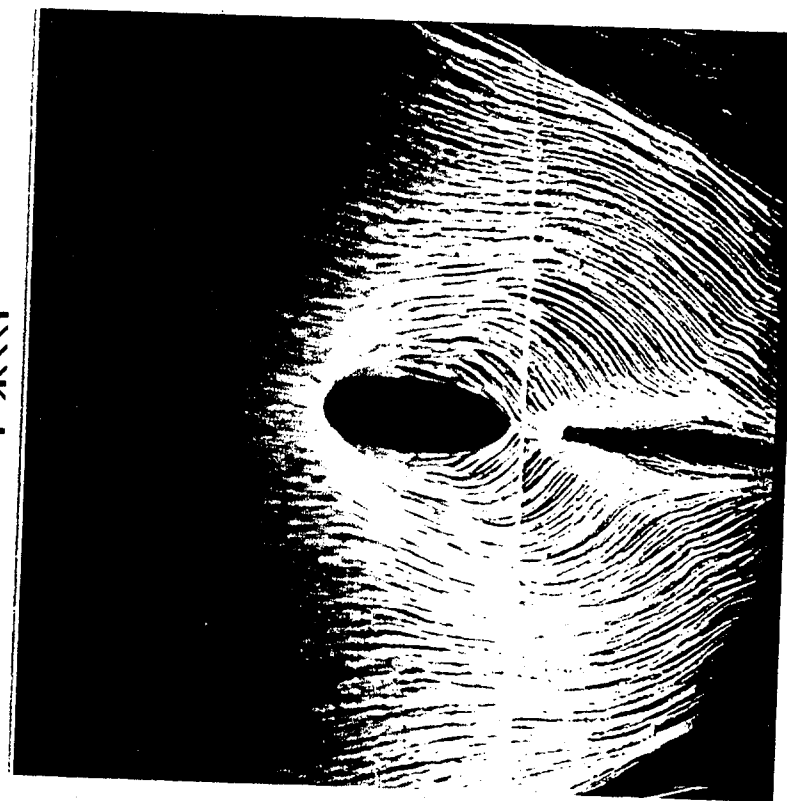
**AERODYNAMICS BRANCH
INTERNAL FLUID MECHANICS DIVISION**



**NASP-HYPERMIXING CONCEPTS
EXPERIMENTAL INVESTIGATION IN 1x1 SWT**



**EXPERIMENTAL CONFIGURATION
BASELINE MODEL**



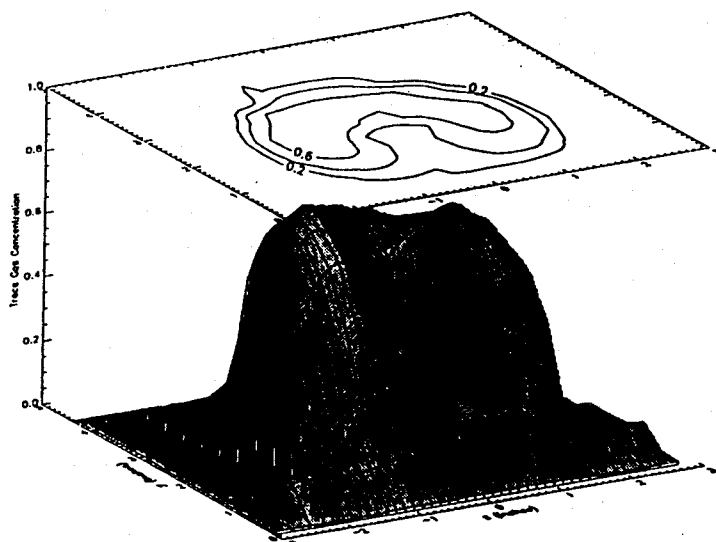
OIL FLOW VISUALIZATION

CD-89-44432

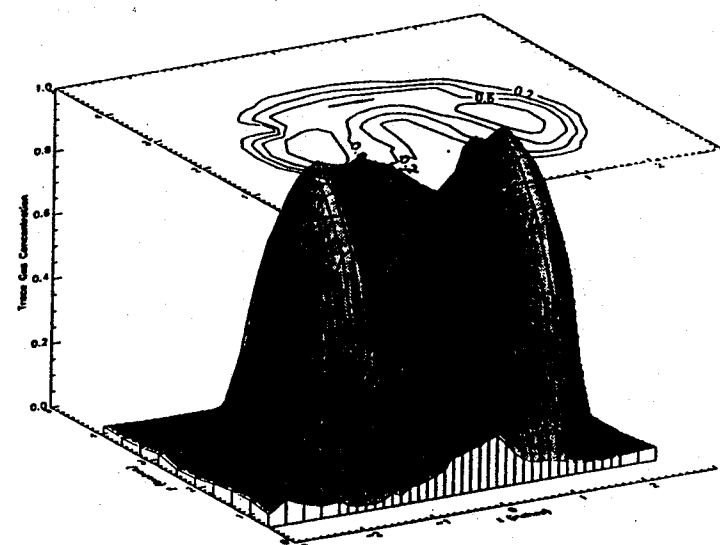
**AERODYNAMICS BRANCH
INTERNAL FLUID MECHANICS DIVISION**



**NASP-HYPERMIXING CONCEPTS
TRACE GAS MEASUREMENTS**



MASS FRACTION—MACH 1.6

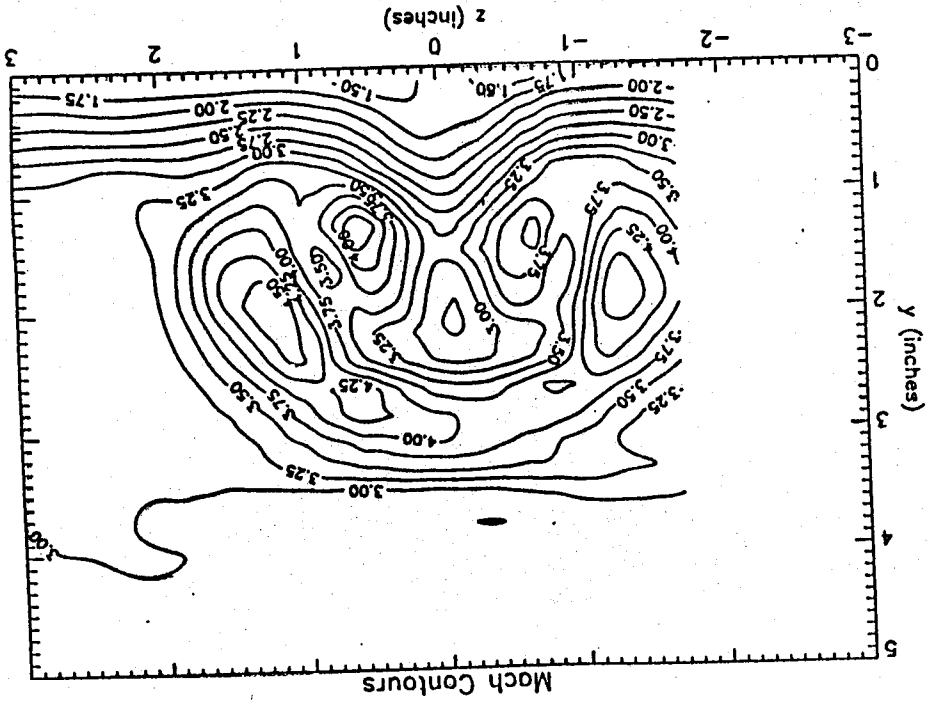


MASS FRACTION—MACH 3.0

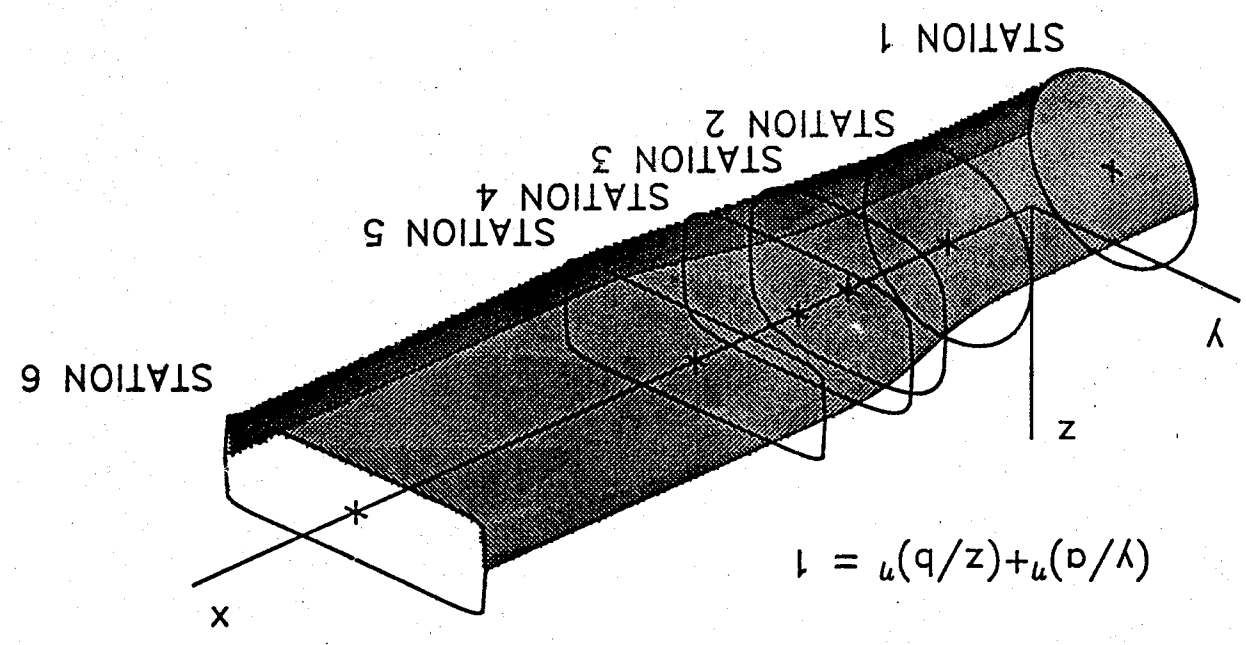
CD-89-44435

102

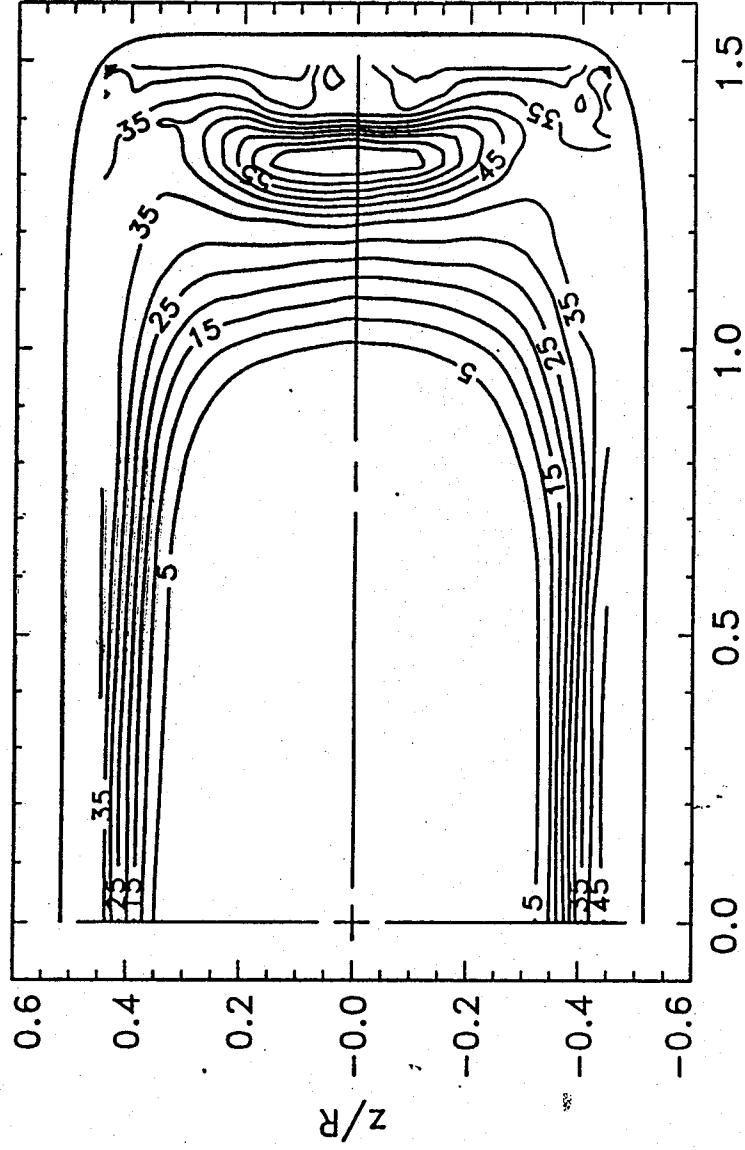
TUNNEL MACH NUMBER - 3.0



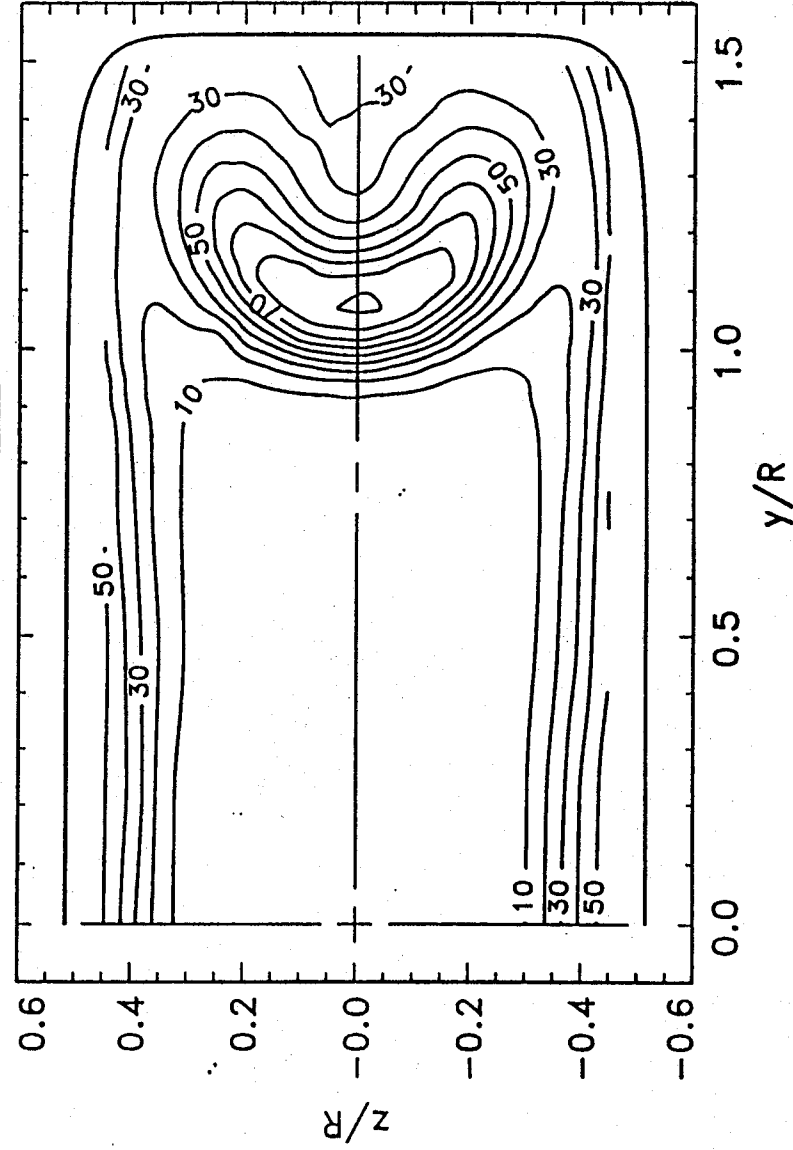
CIRCULAR-TO-RECTANGULAR TRANSITION DUCT



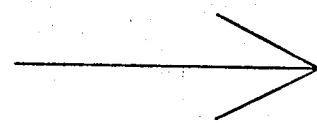
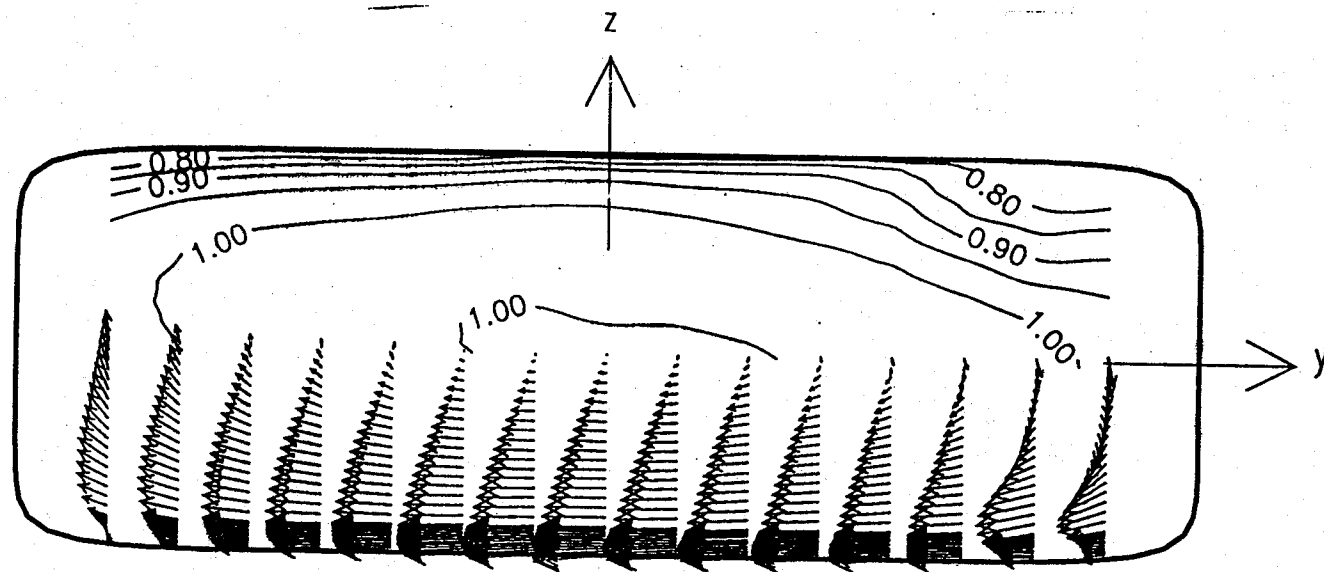
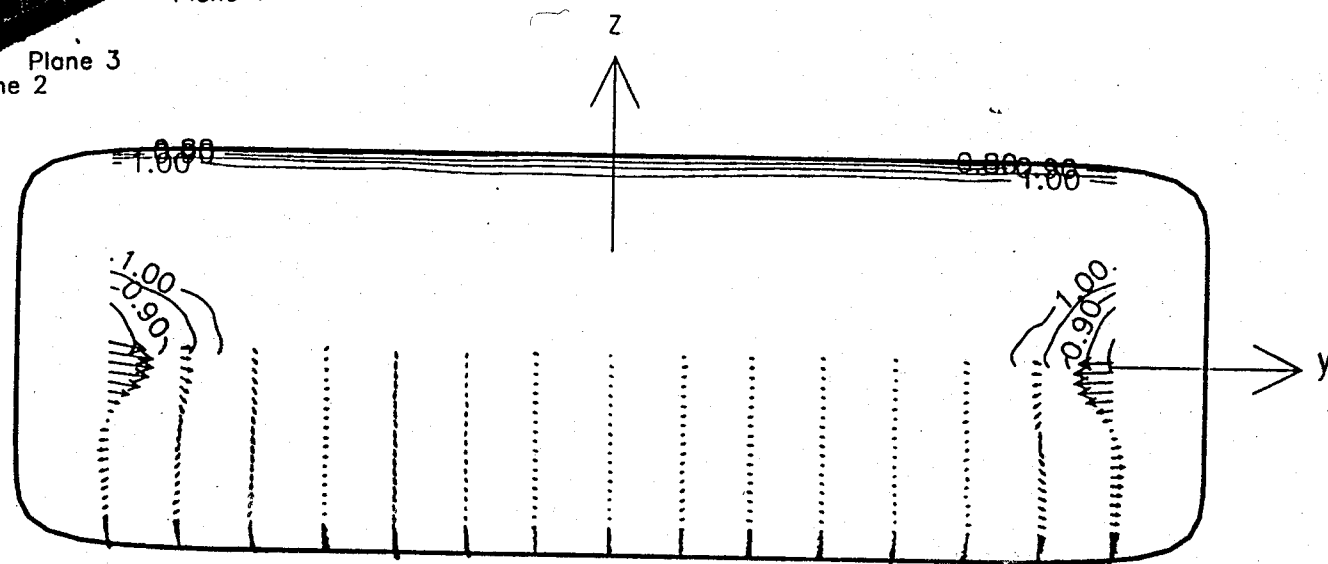
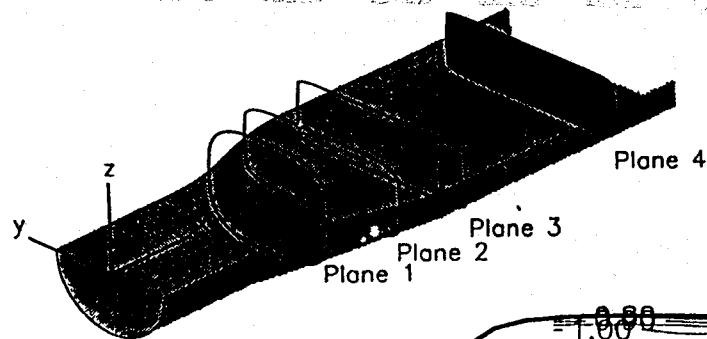
STATION 5



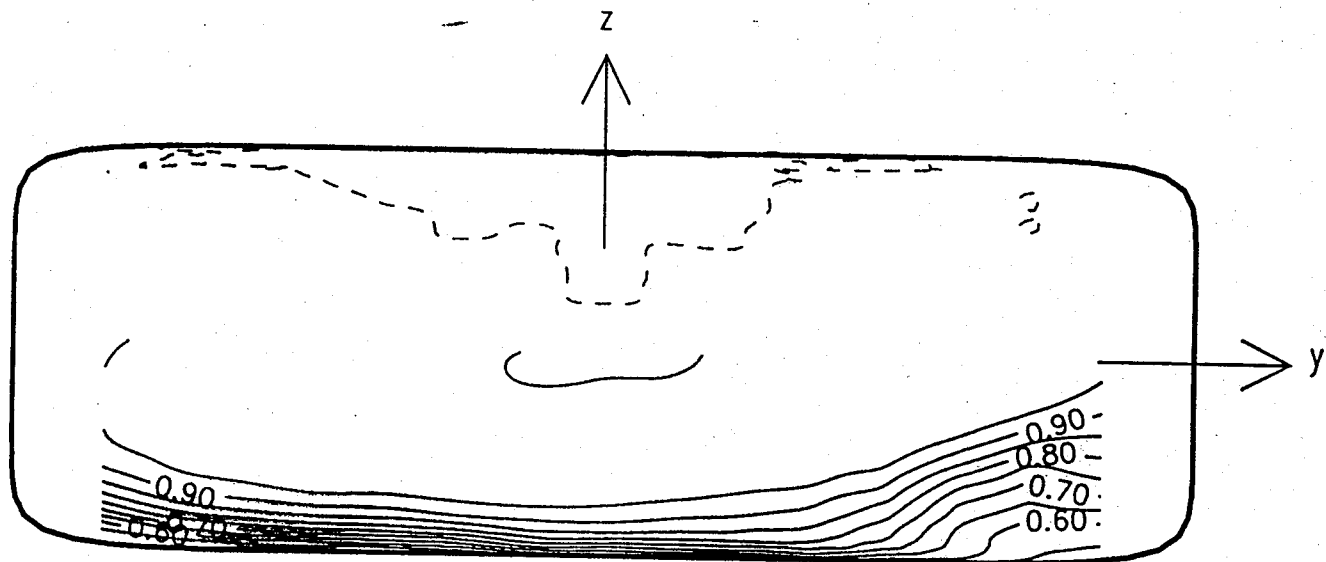
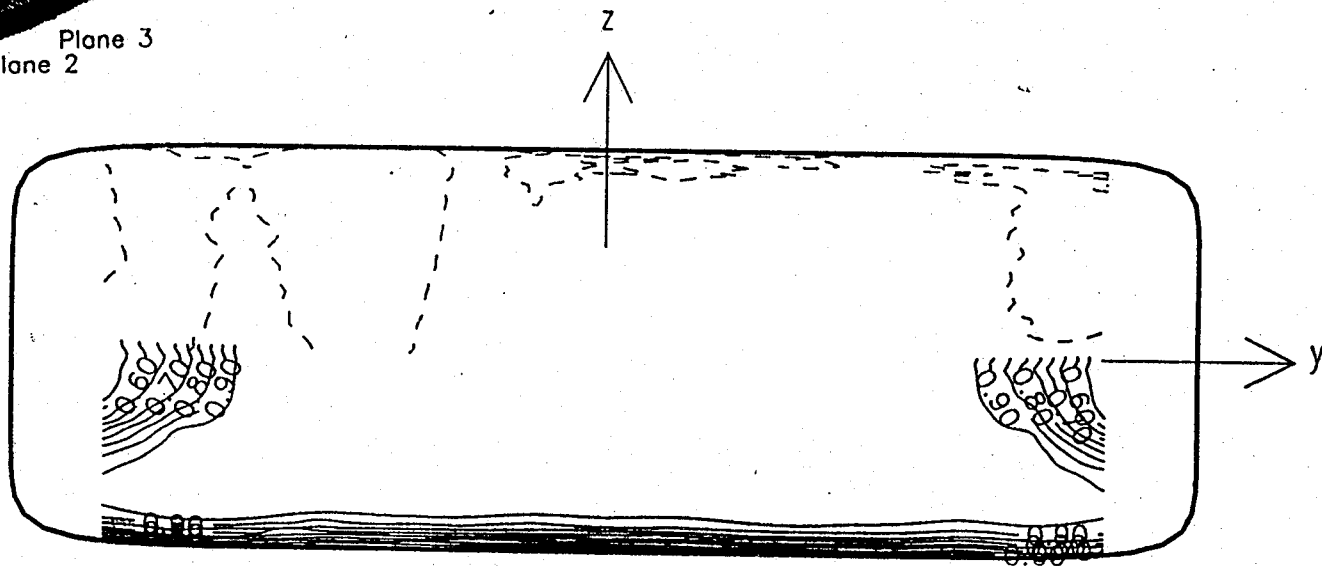
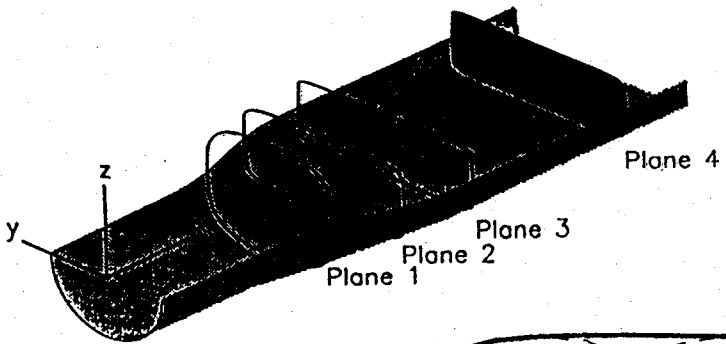
STATION 6

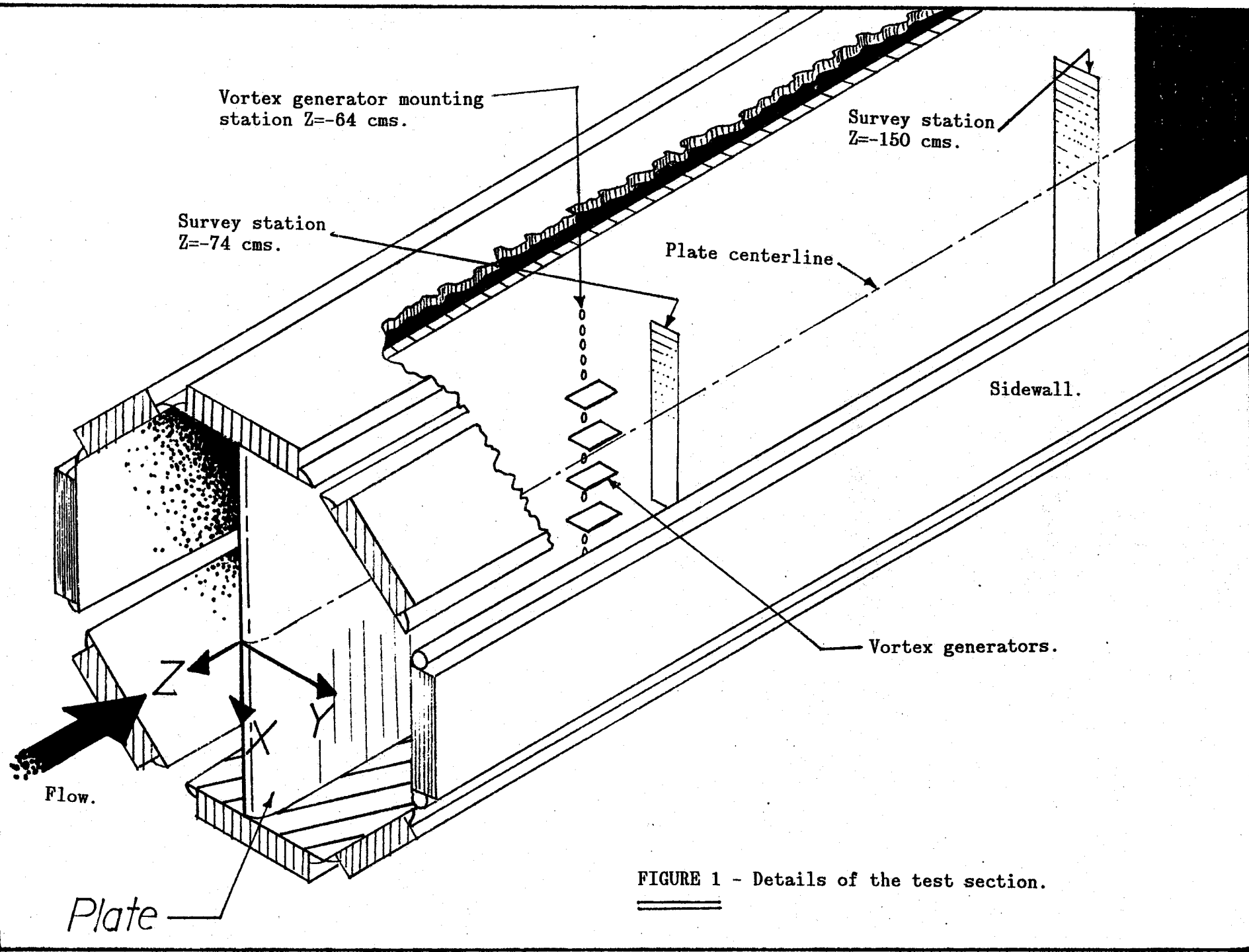


TURBULENCE KINETIC ENERGY CONTOURS
 $(k/U_b^2) \times 10^4$



Inlet Mach
204





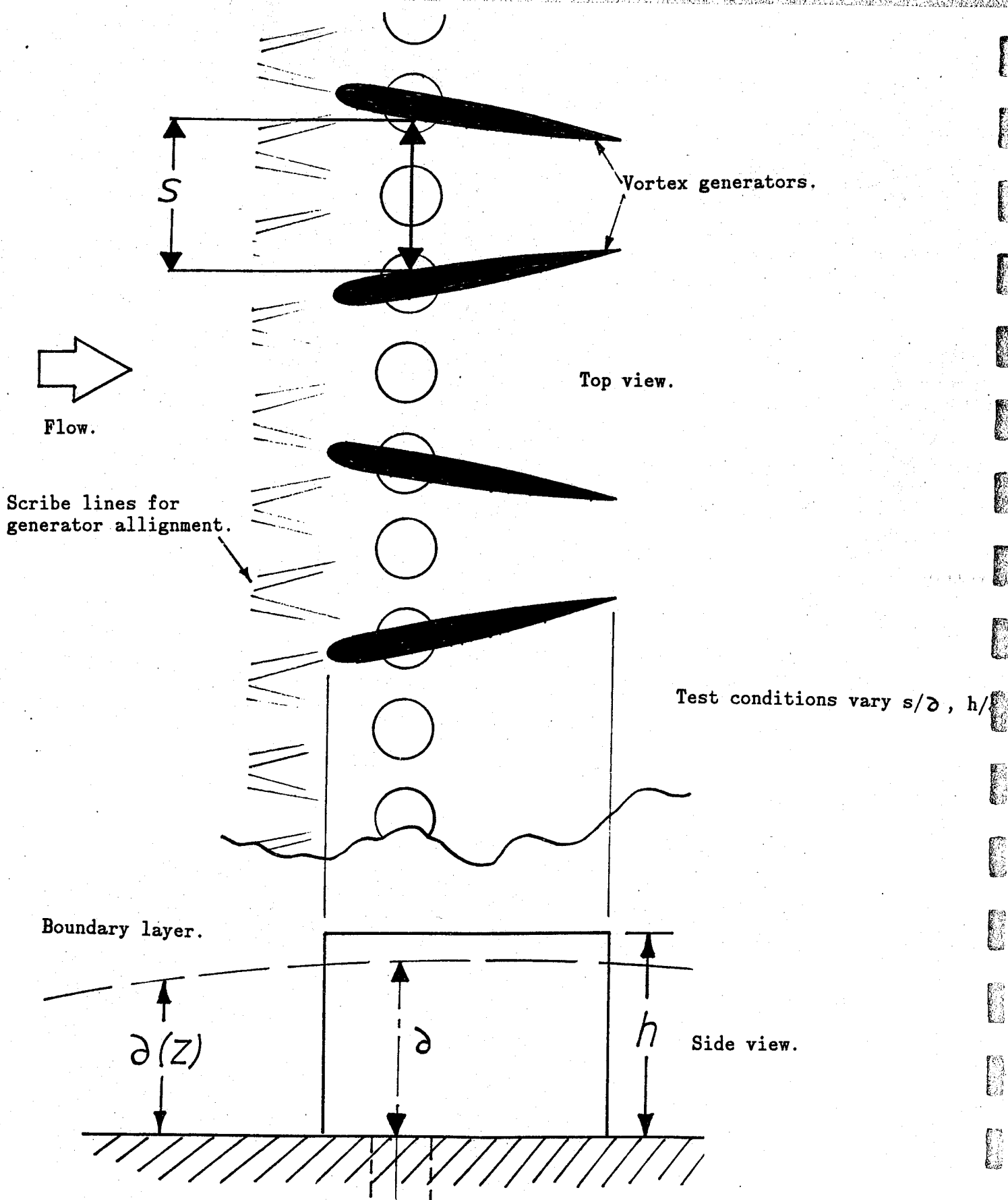


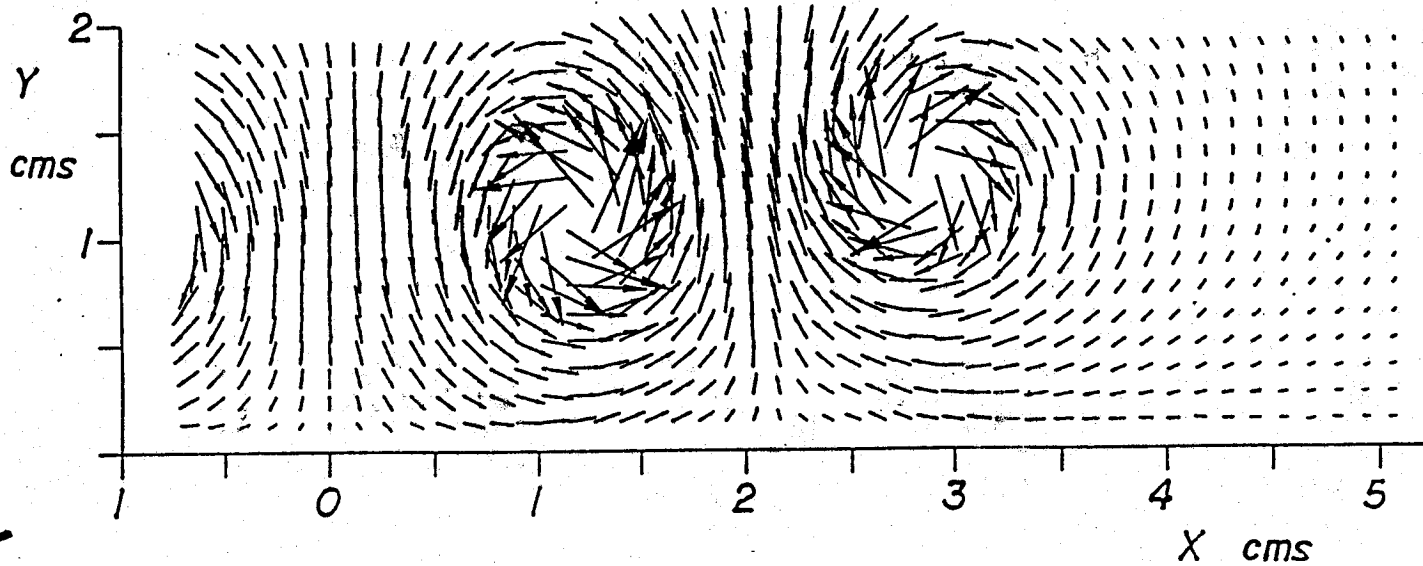
FIGURE 2 - Details of the vortex generator mounting station.

SECONDARY VELOCITY VECTORS

Station 74 - Embedded vortex array, spacing ratio = 2.0.

DATA

46x15

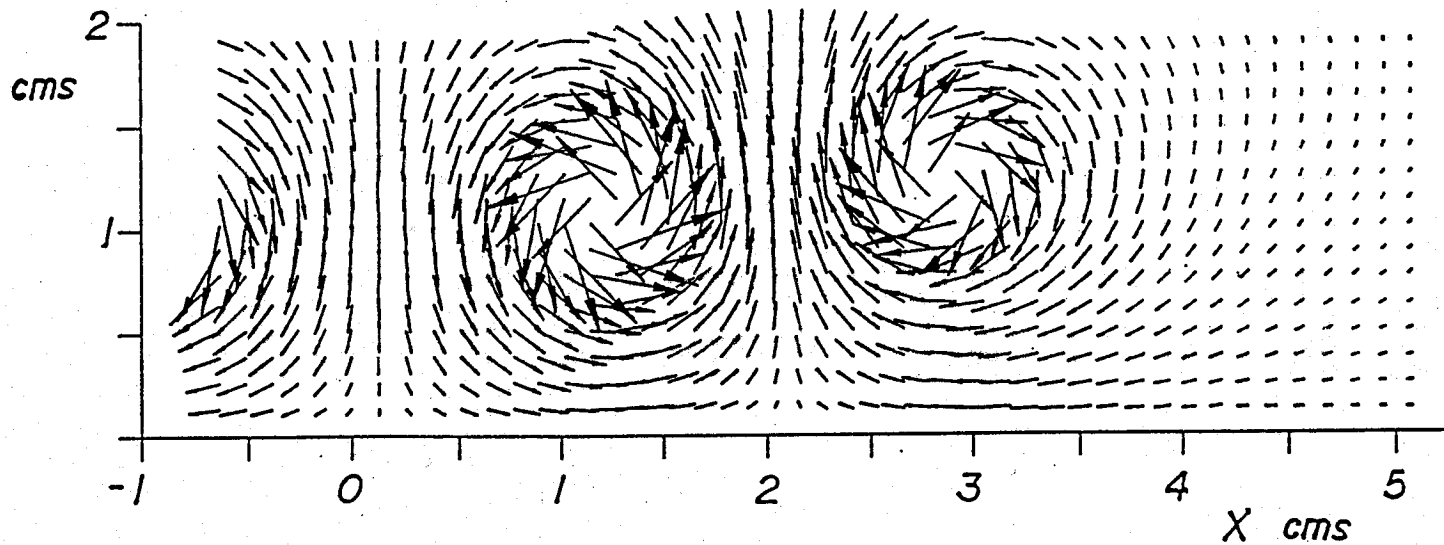


SECONDARY VELOCITY VECTORS

Oseen model of the data for a spacing ratio of 2.0.

Model

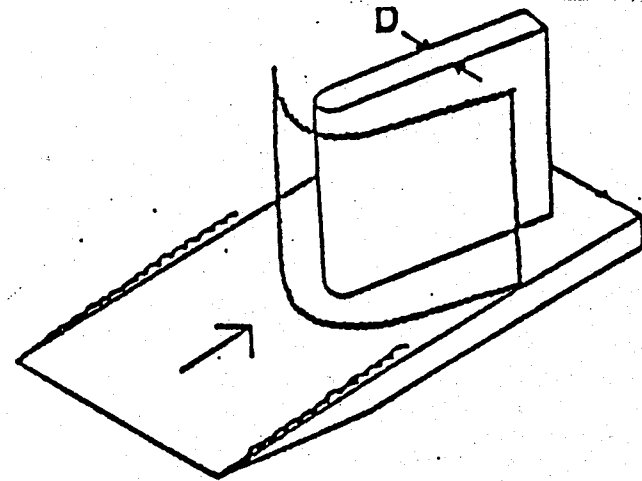
46x15



Vortex array embedded in a boundary layer. Top picture is the data taken at Mach .2. Bottom picture shows a model of the data composed of Oseen vortices with vortex core locations, circulations, and peak vorticities corresponding to the data results.

UNSTEADY SHOCK/BOUNDARY-LAYER INTERACTION

- UNIVERSITY OF TEXAS AT AUSTIN
MACH 5 (DOLLING)
- PRINCETON UNIVERSITY
MACH 3 (SMITS)
- LeRC 1X1 SWT
MACH 3 AND 5 (BARNHART, HINGST)
- FLIGHT TESTING ?



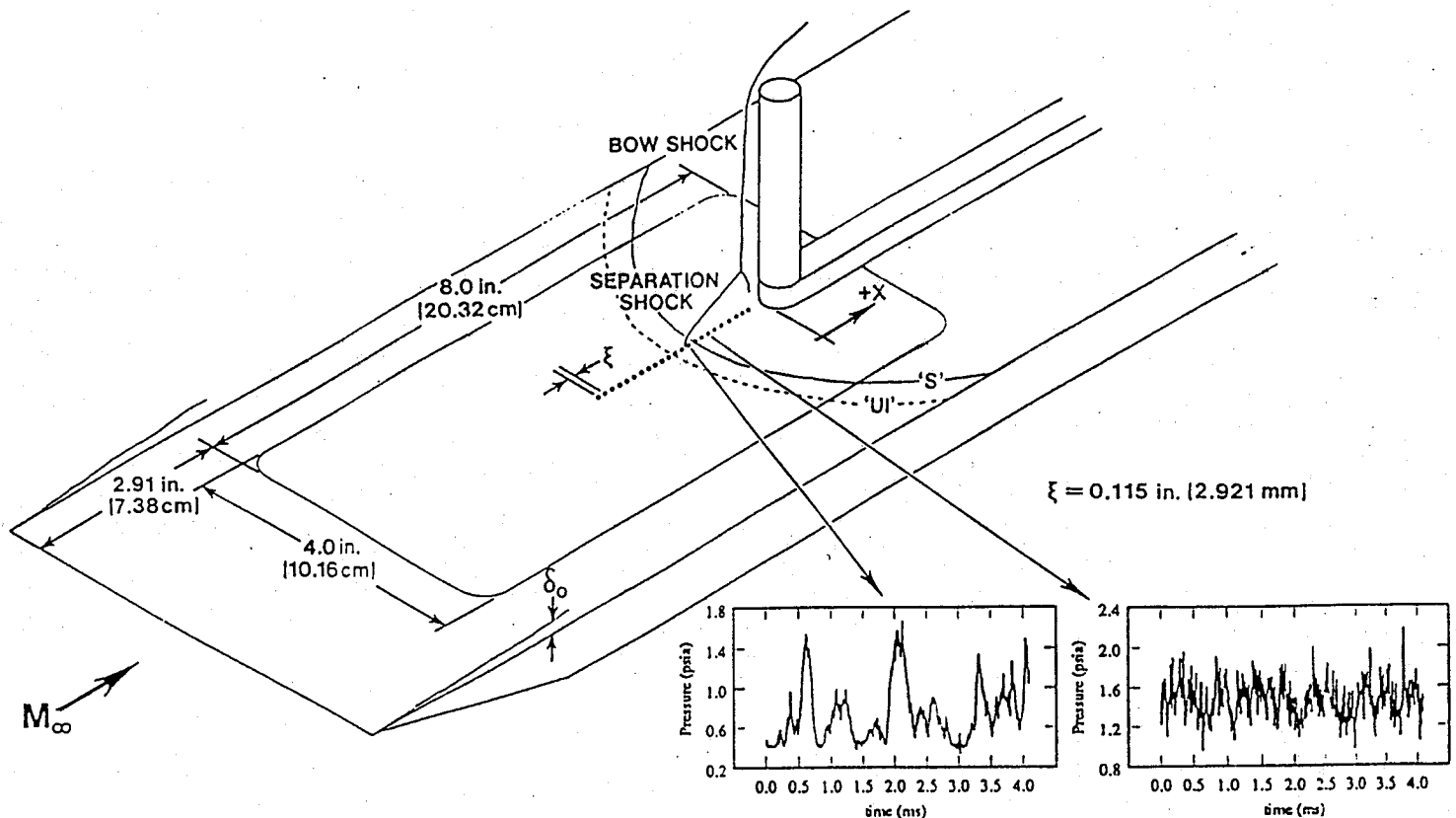
Dynamics of Shock Wave Turbulent Boundary Layer Interactions Induced by Cylinders and Blunt Fins

Principal

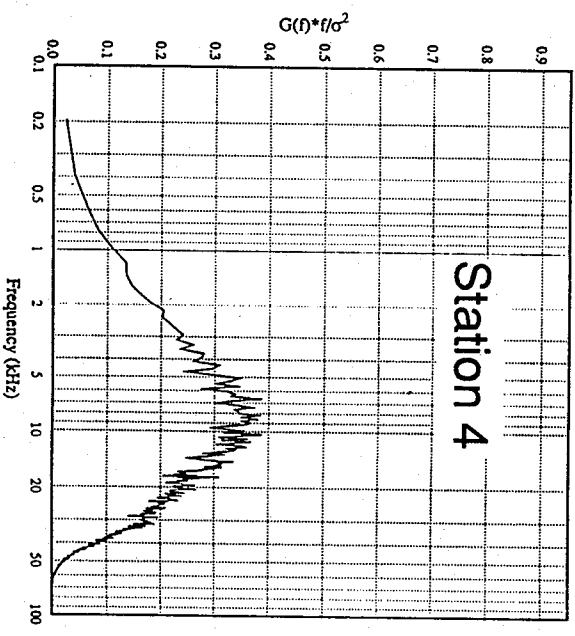
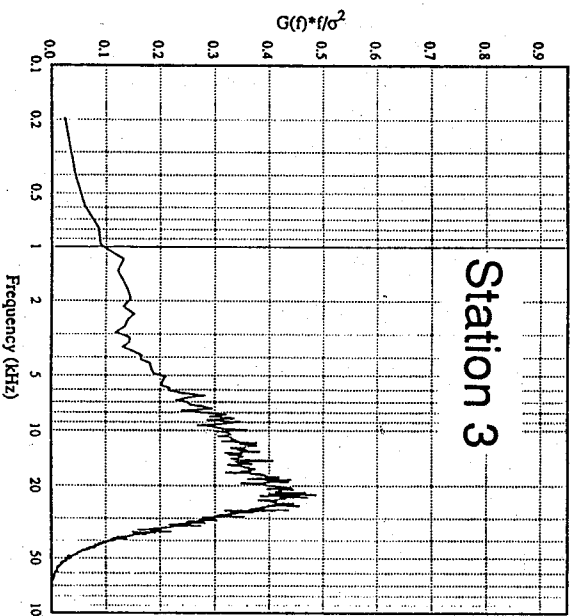
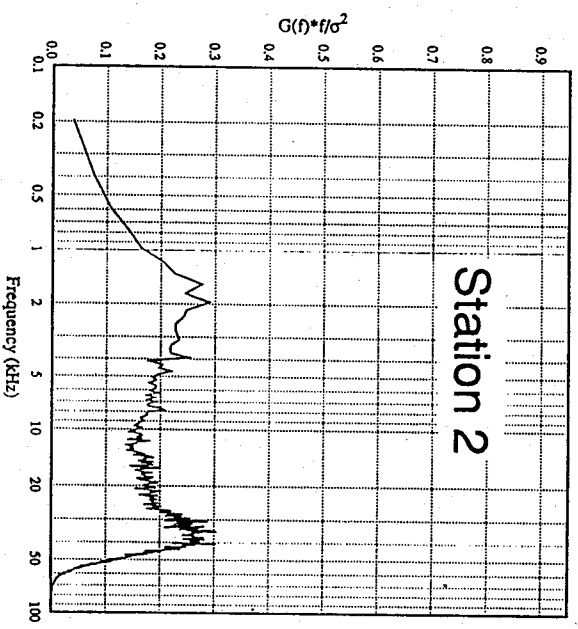
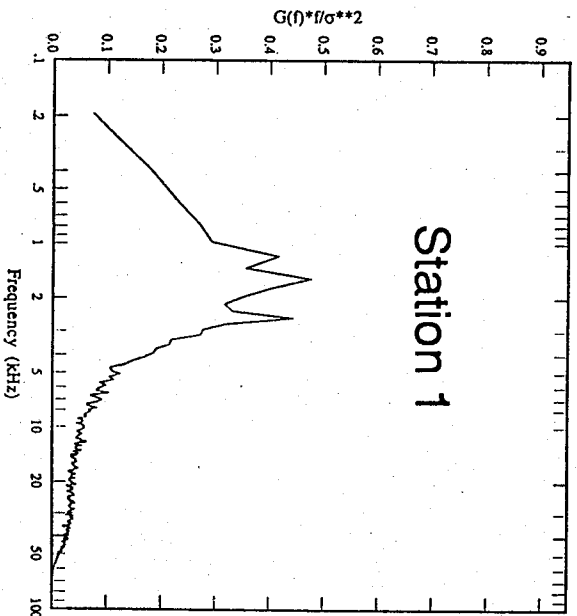
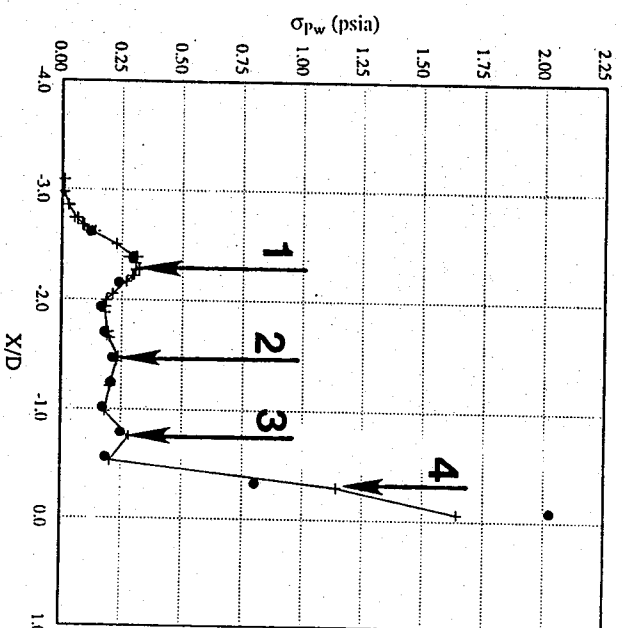
Investigator: D. S. Dolling, Aerospace Engineering
and Engineering Mechanics,
The University of Texas at Austin

Graduate Research Asst.: Leon Brusniak

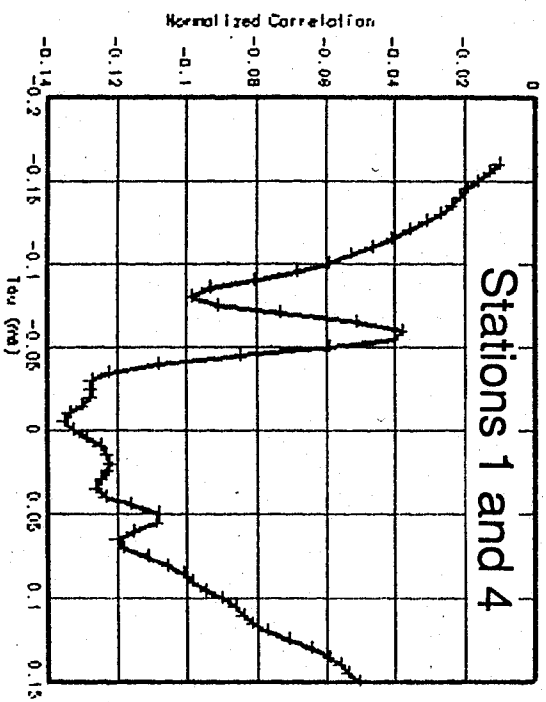
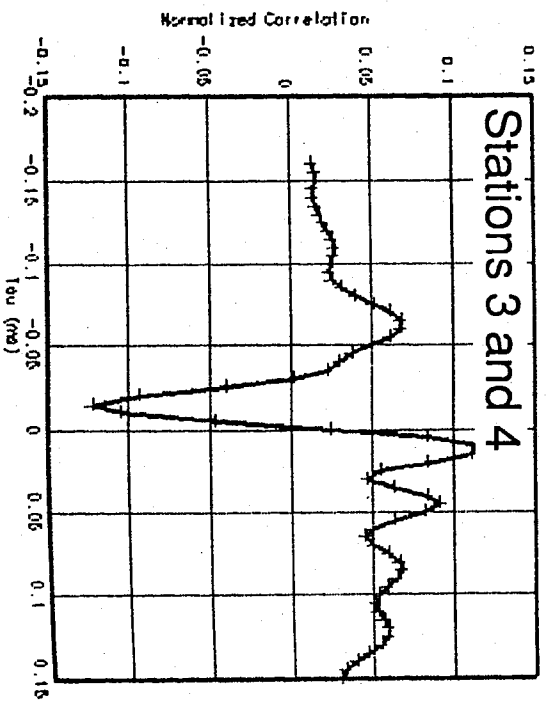
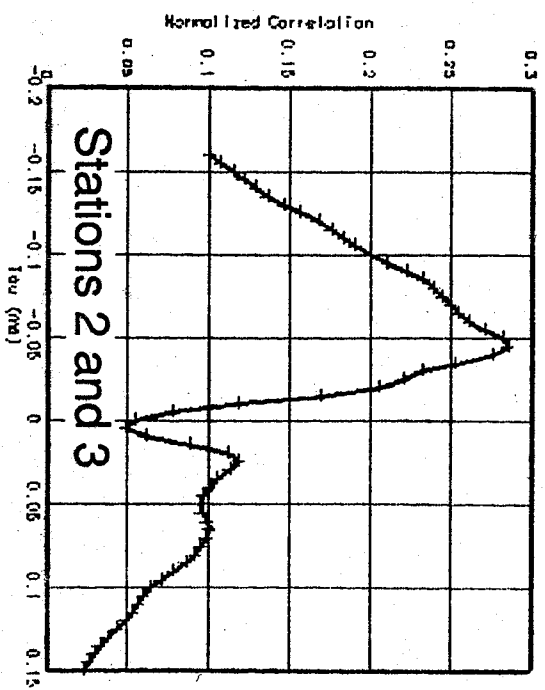
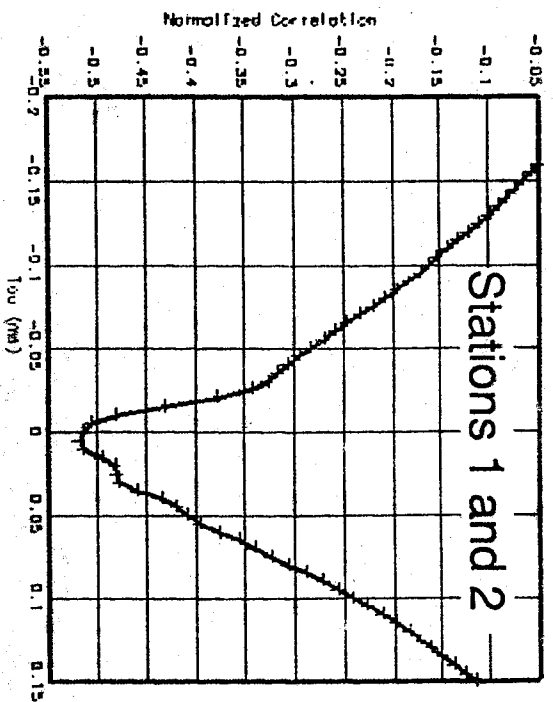
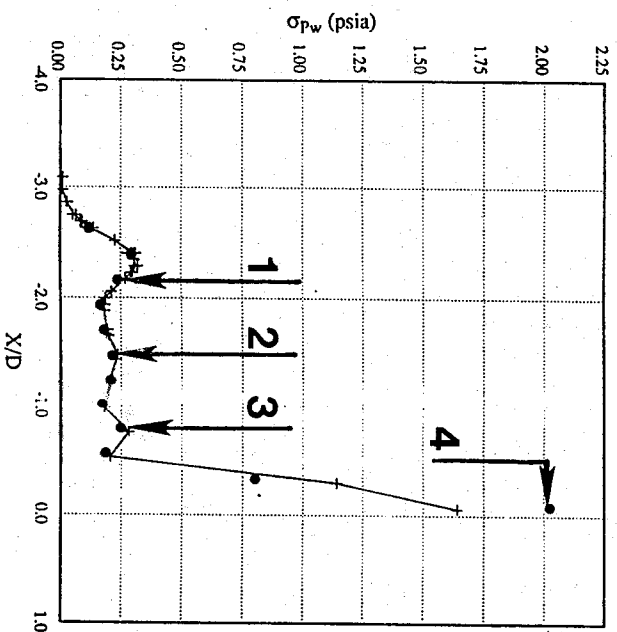
Sponsor: NASA Lewis Research Center
(Warren Hingst)



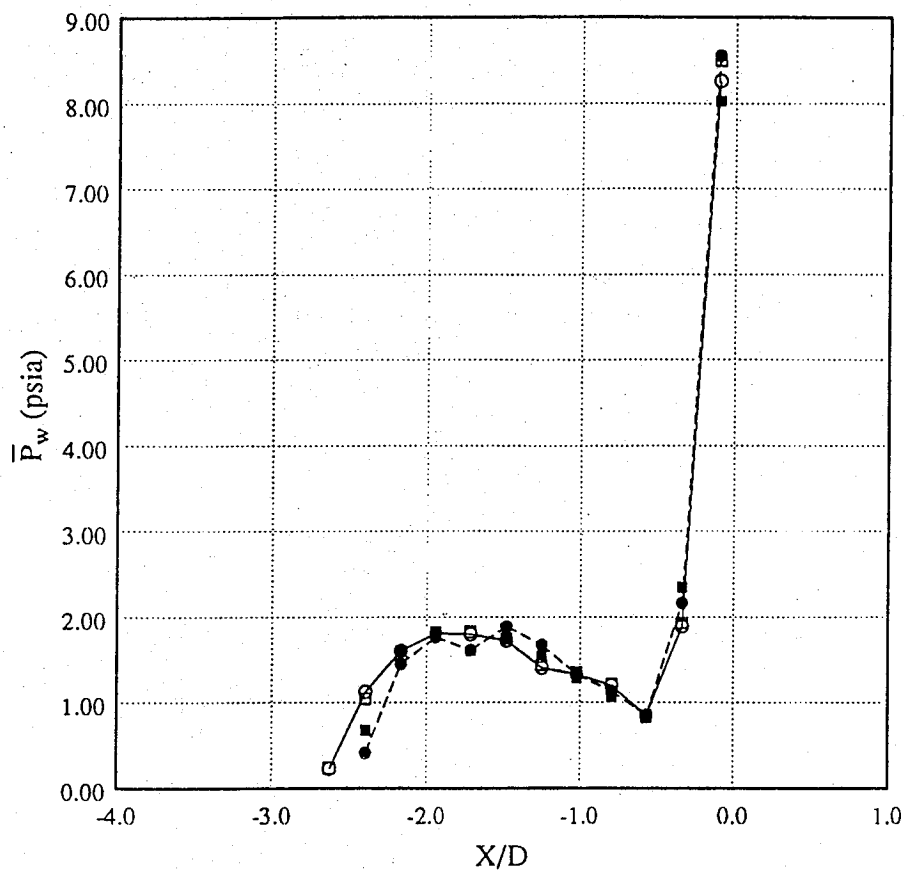
POWER SPECTRA



CROSS-CORRELATION



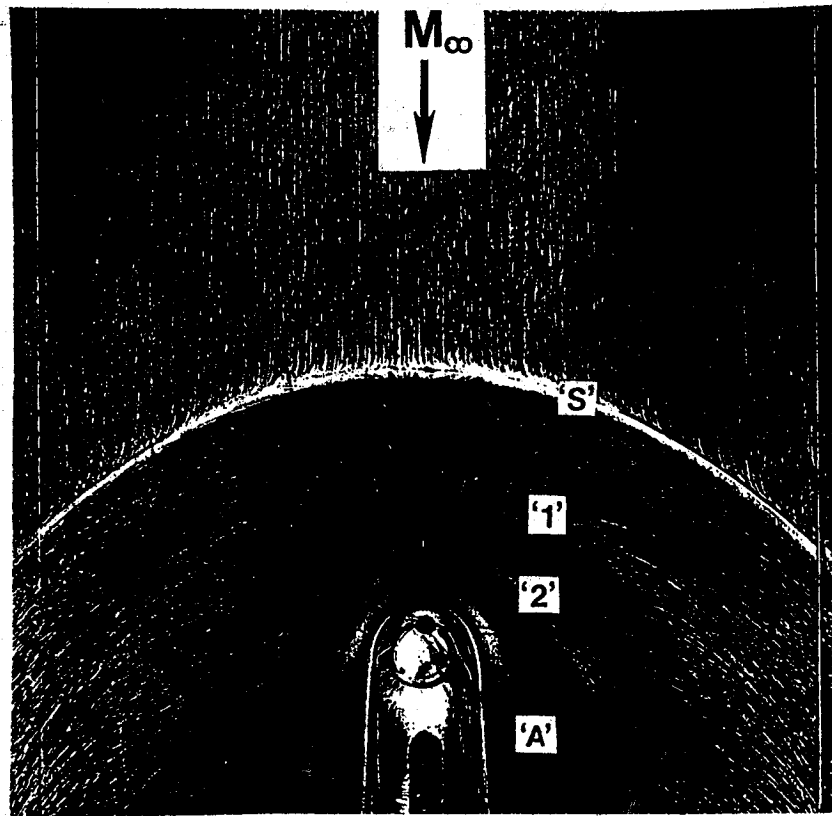
Ensemble-Averaged Wall Pressure Distribution on Centerline Upstream of Cylinder ($D = 1/2''$)



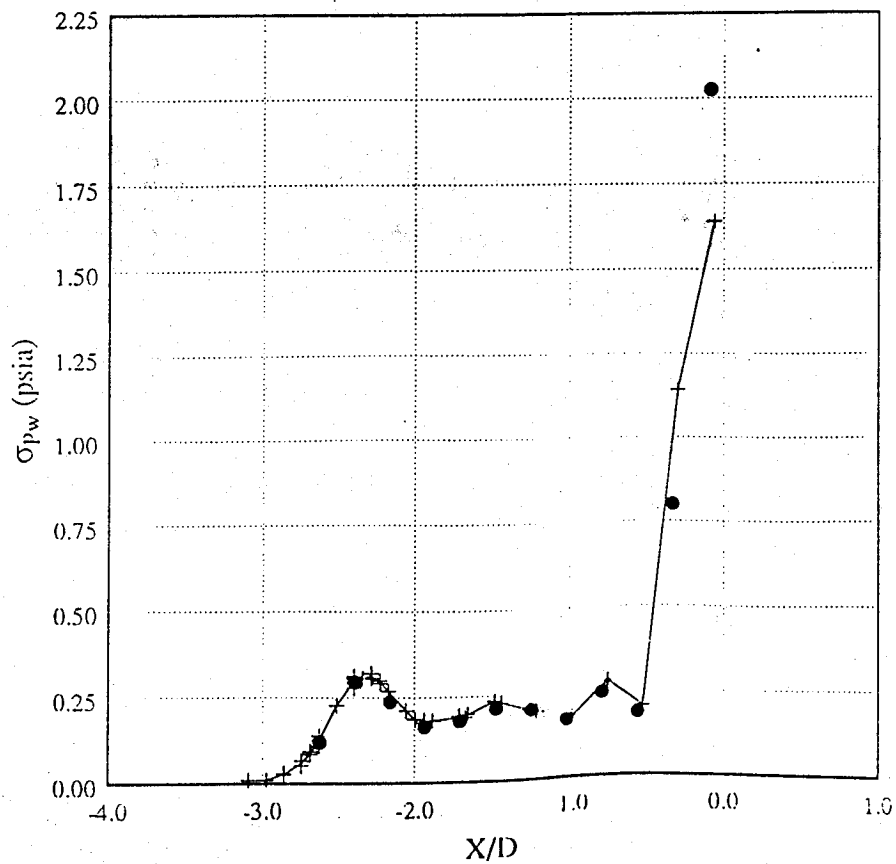
Open Symbols: Shock-forward, $\gamma \approx 0.2$

Closed Symbols: Shock-back, $\gamma \approx 0.8$

Surface Flow Pattern



RMS Distribution on Centerline



Flow Conditions:

$$Ma_e = 2.9$$

$$T_o = 265 \text{ } ^\circ\text{K}$$

$$U_e = 575 \text{ m/s}$$

$$Re_e/m = 6.5 \times 10^7$$

Blunt fin diameter: 0.75"

Boundary layer thickness: 1.1"

Visualization:

- * Nd:YAG laser with a fourth harmonic generator operating at 266 nm (far UV), 4 ns pulses at 10 Hz.
- * Images obtained with a an intensified, UV sensitive CID camera.
- * Flow is from right to left

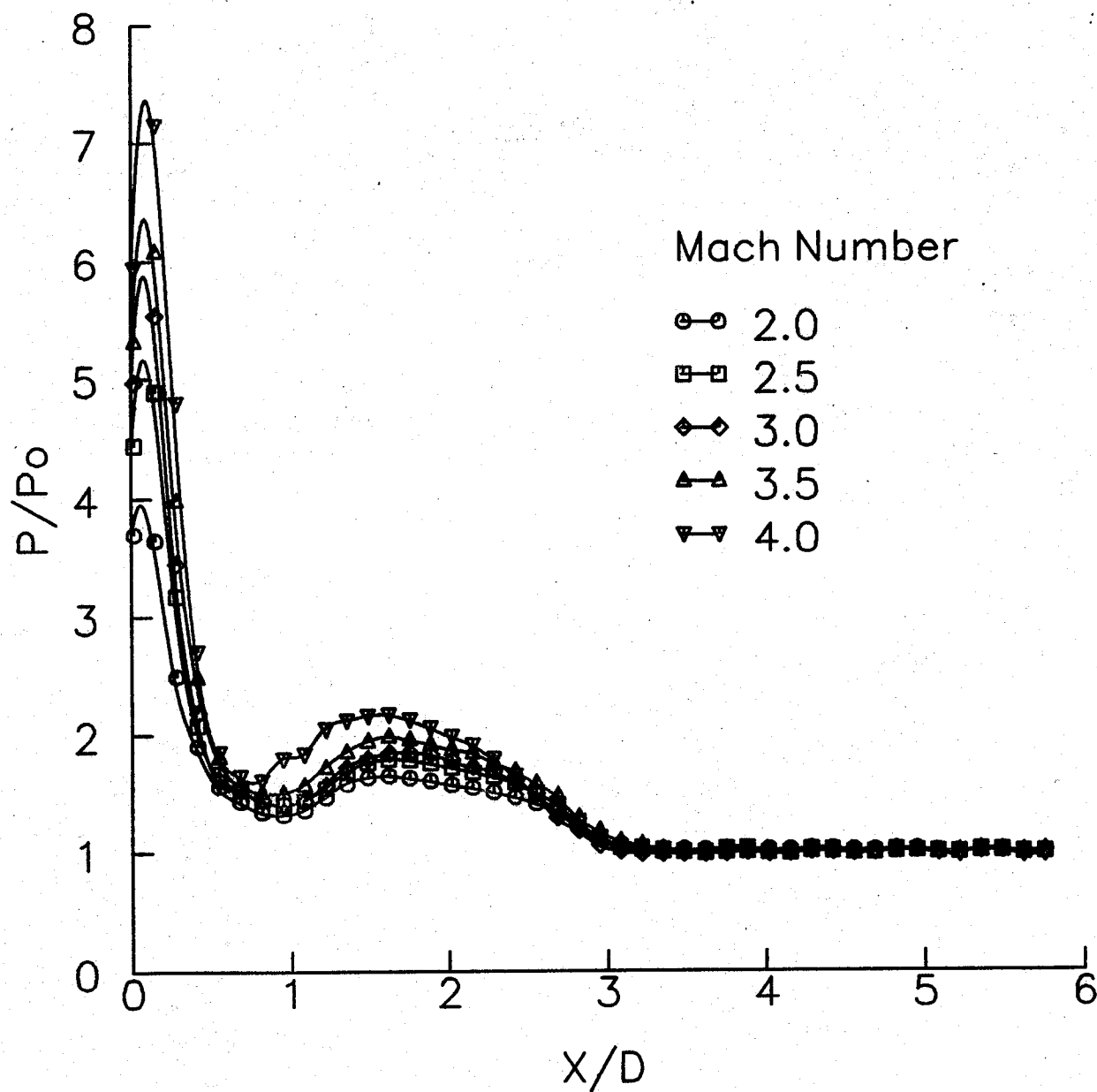
Rayleigh scattering

- * In air flows: small ice clusters (30 nm) can dominate the Rayleigh signal. Yields high intensity signals in regions of low temperature, and signal dropout in regions of high temperature. A strong shock will appear as the boundary between a bright region of low density fluid and a dark region of high density fluid (the reverse of the expected result).
- * In nitrogen flows: the Rayleigh signal is now dominated by scattering from nitrogen molecules. A shock will appear as the boundary between a darker region of low density fluid and a brighter region of high density fluid (as expected).

Rayleigh Scattering:

- * scattering by particles much smaller than the wavelength of light.
- * The intensity of the signal is proportional to the density of the scattering centers - gives the possibility of quantitative density maps.
- * Can reveal instantaneous turbulence and shock structure in a plane.
- * Sensitivity proportional to the fourth power of frequency, so operation at short wavelengths very beneficial.

Blunt Fin Interaction Static Pressures



ORIGINAL PAGE IS
OF POOR QUALITY

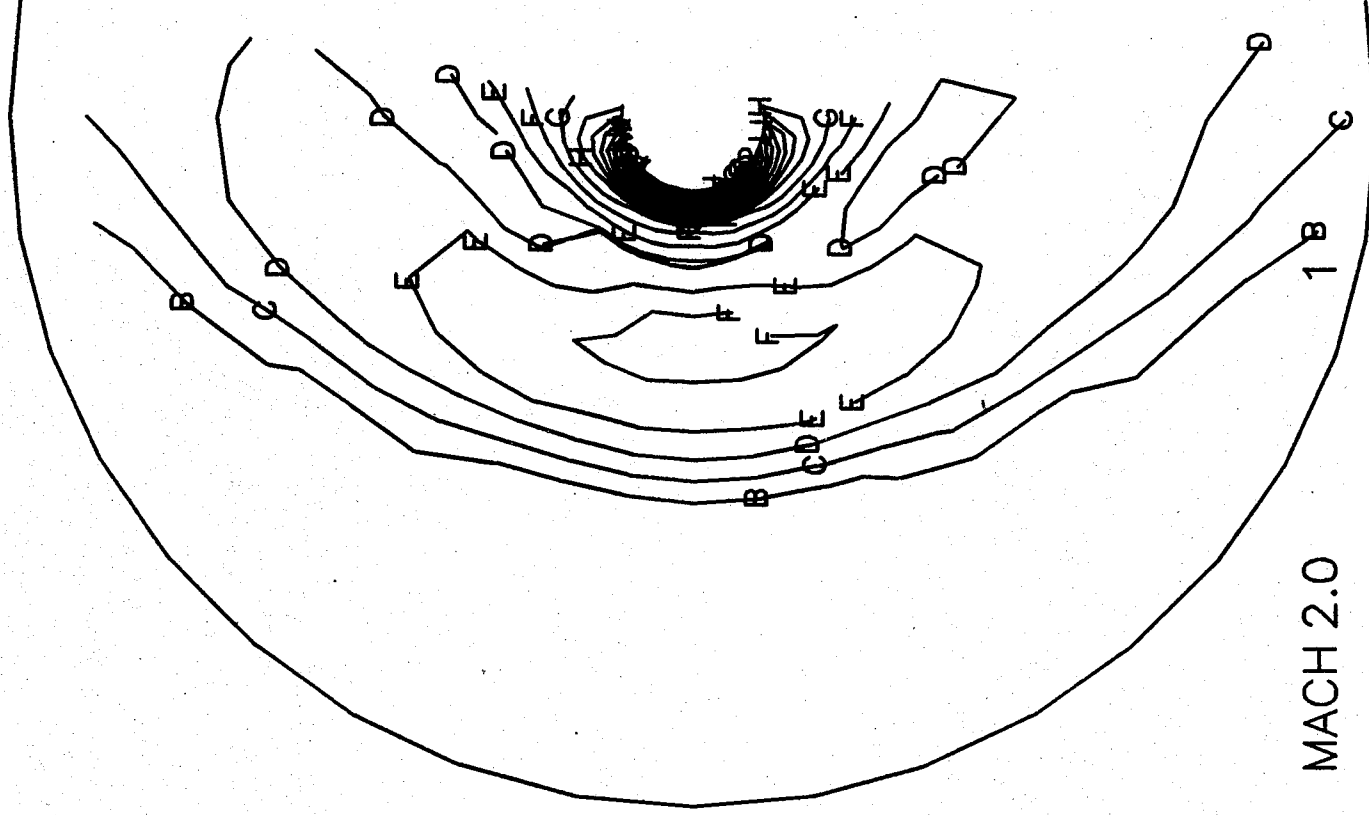


ORIGINAL PAGE IS
OF POOR QUALITY



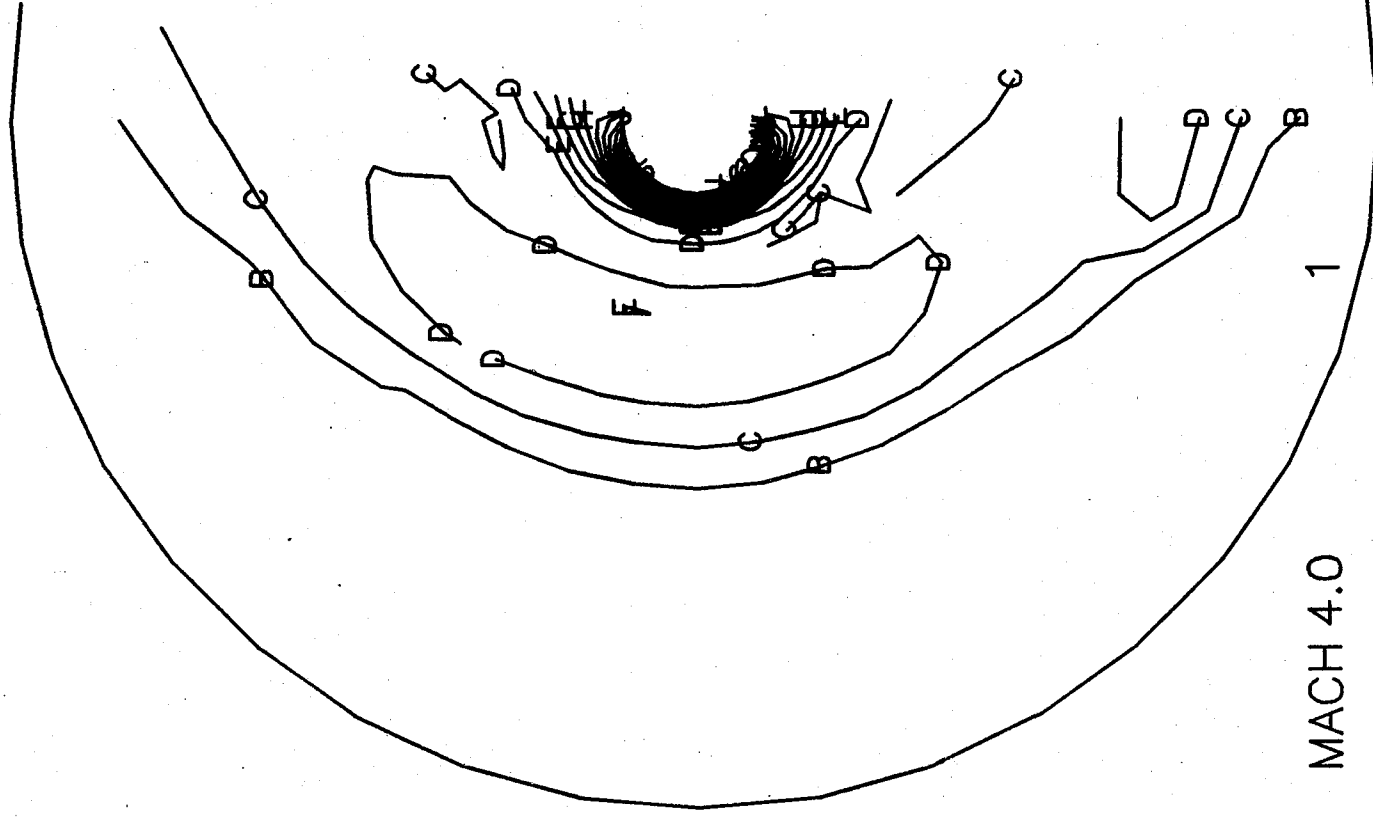
2.6987
3.0252
3.3516
3.6781
4.0045
4.3310
4.6574
4.9839
5.3103
5.6368
5.9632
6.2897
6.6161
6.9426
7.2690
7.5955
7.9219
8.2484
8.5748
8.9013
9.2277

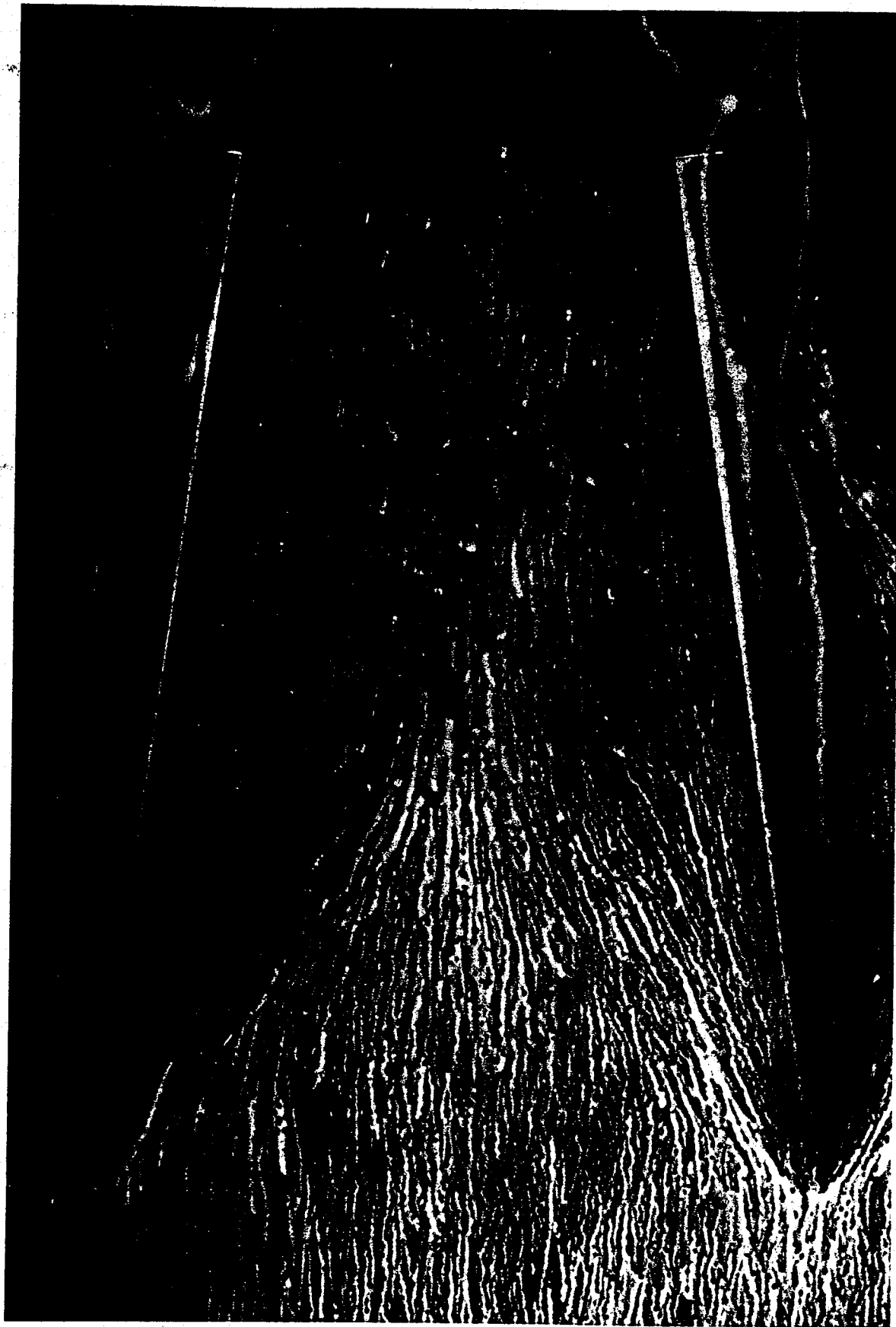
A B C D E F G H I J K L M N O P Q R S T U

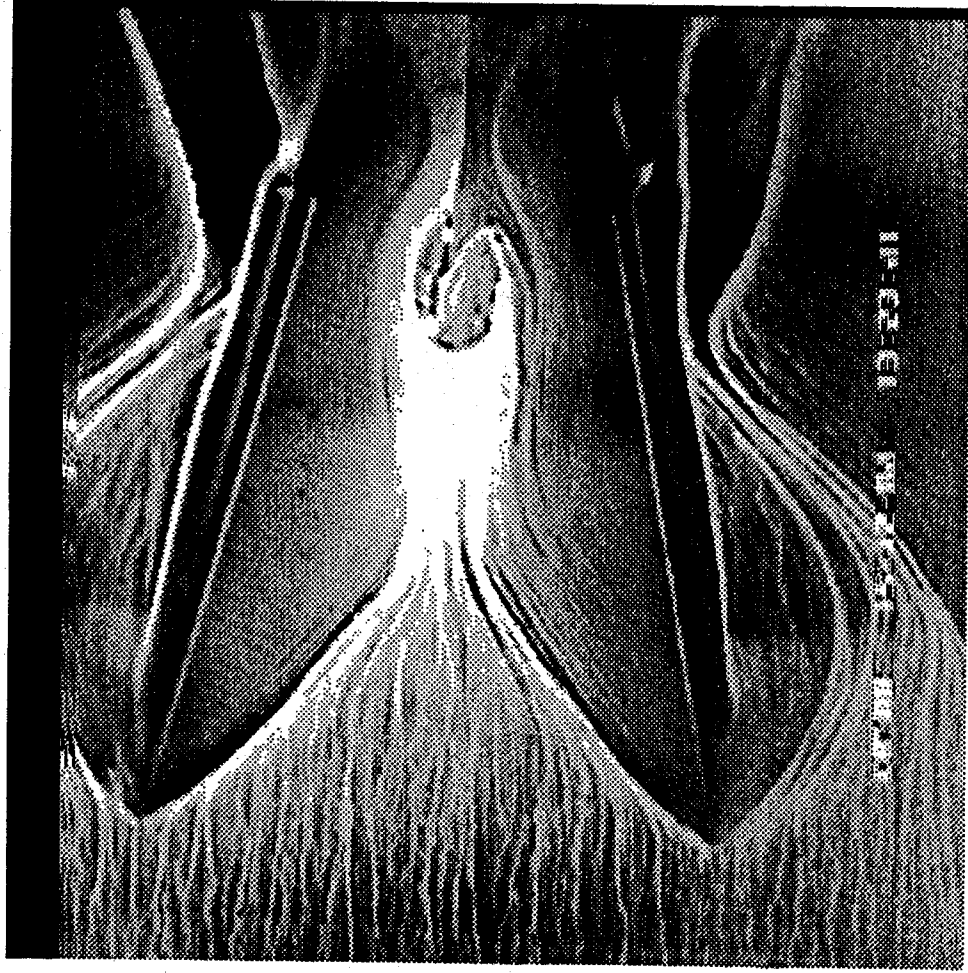


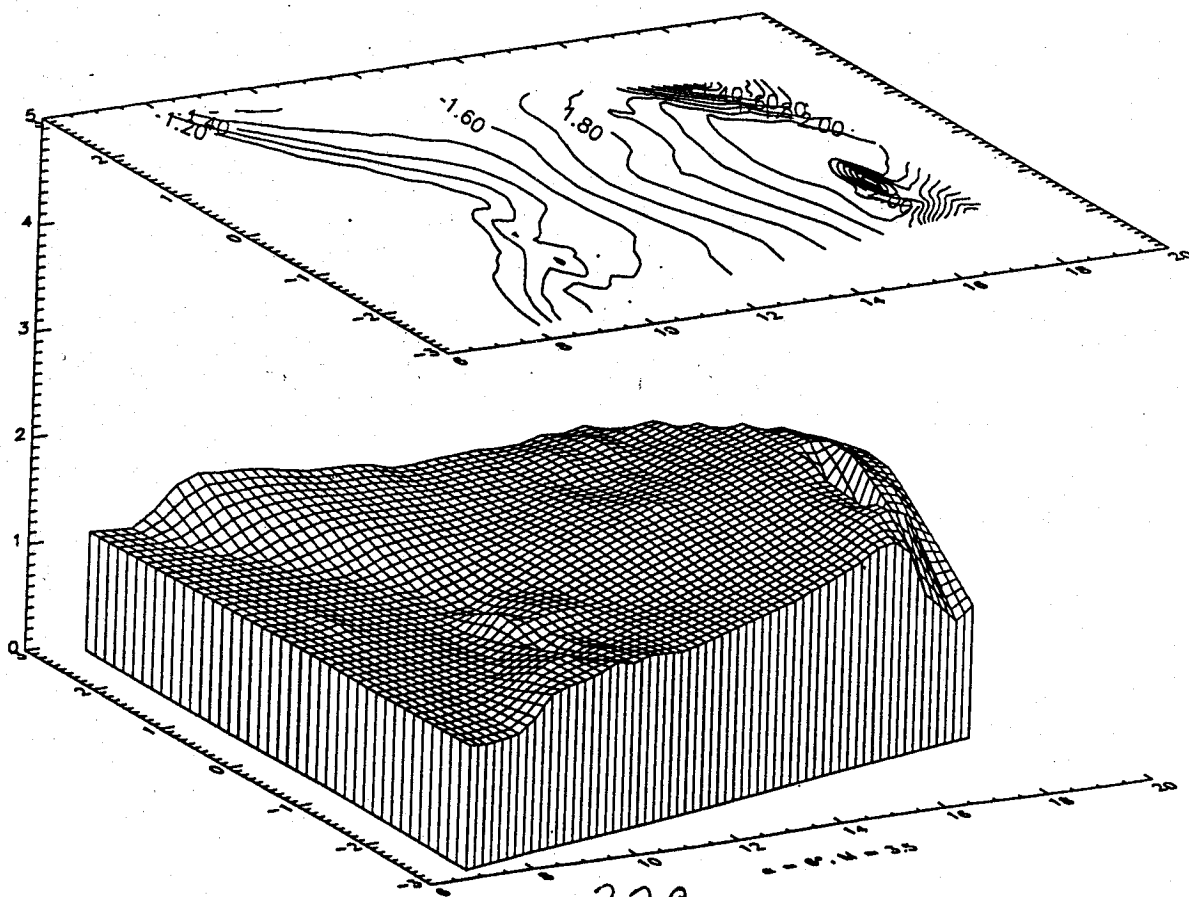
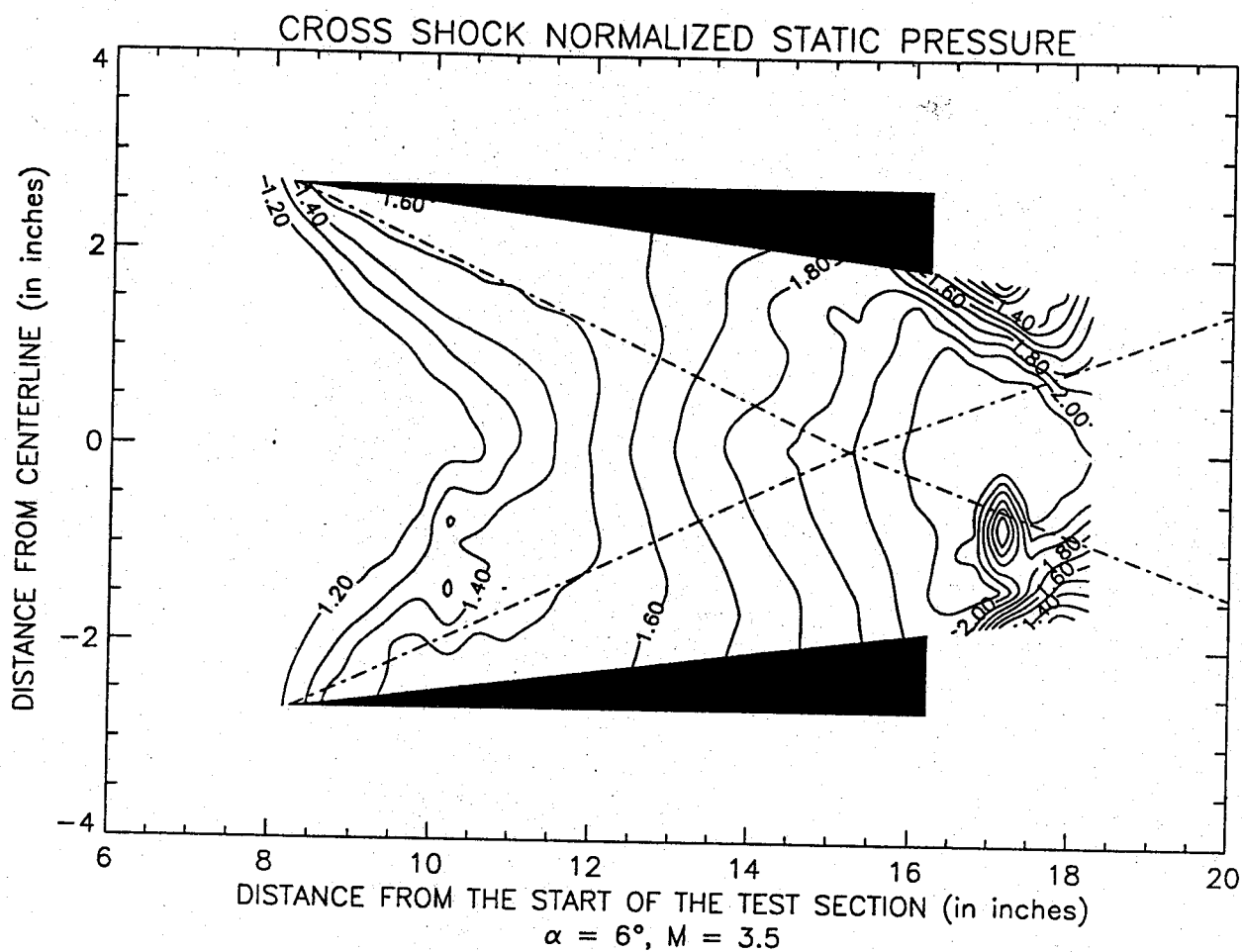
0.2730
0.3531
0.4332
0.5134
0.5935
0.6736
0.7537
0.8339
0.9140
0.9941
1.0742
1.1544
1.2345
1.3146
1.3947
1.4749
1.5550
1.6351
1.7152
1.7954
1.8755

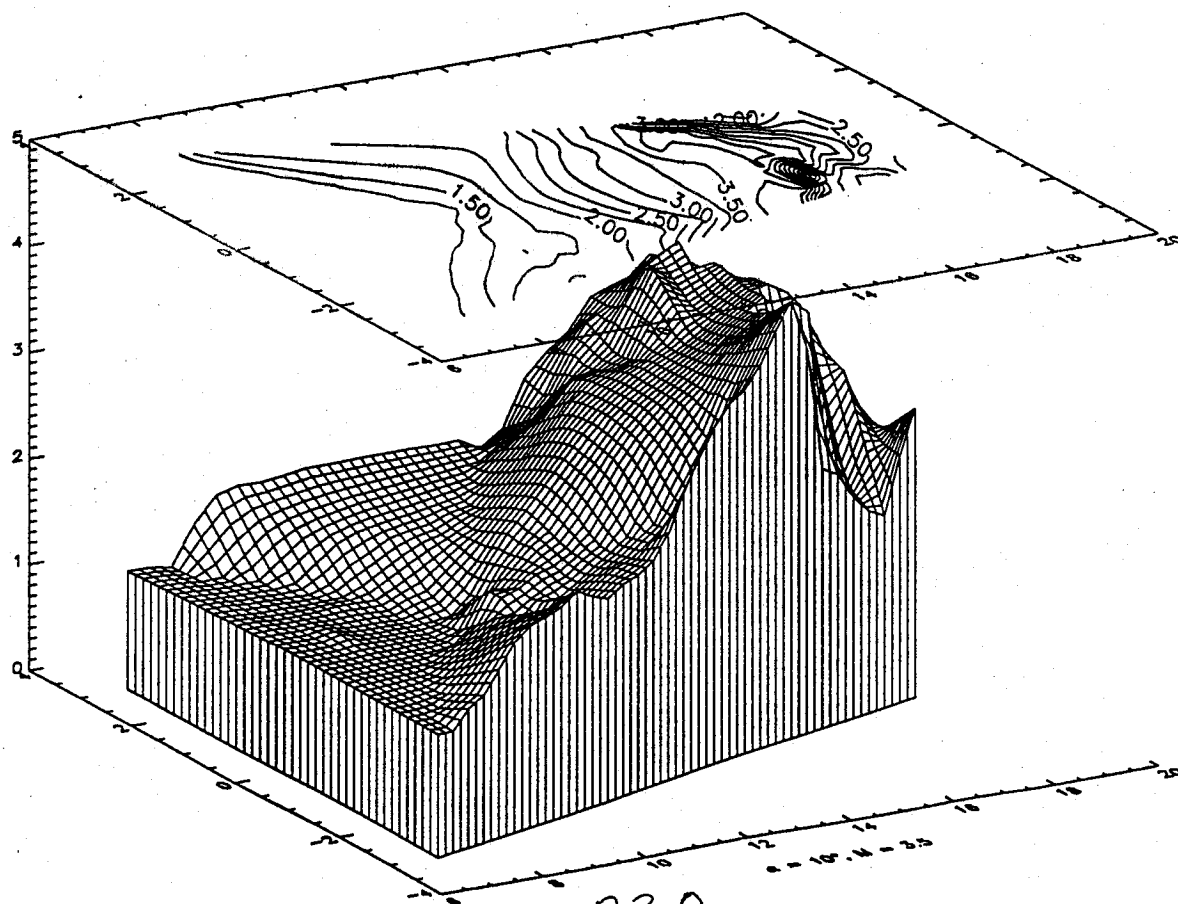
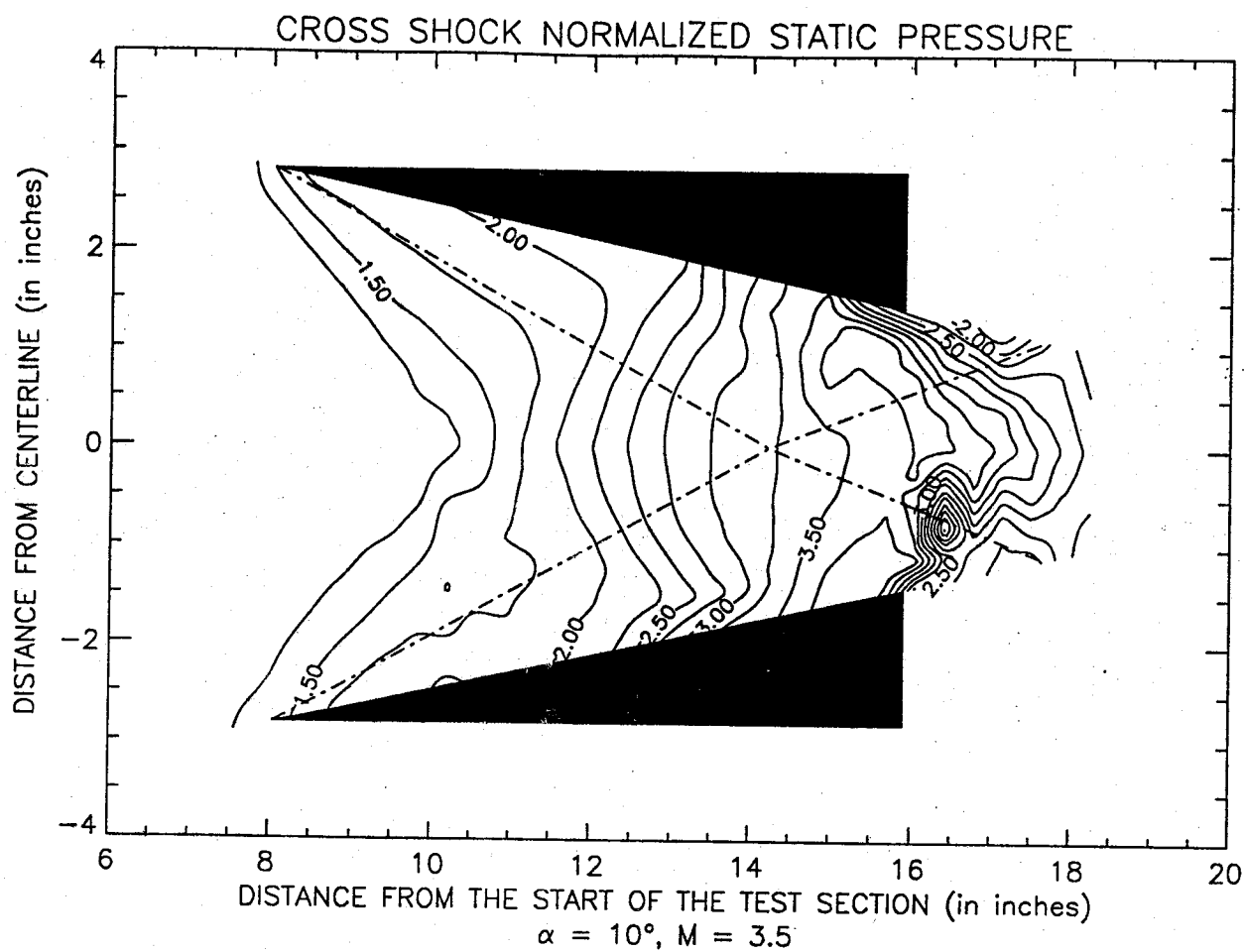
A B C D E F G H I J K L M N O P Q R S T U



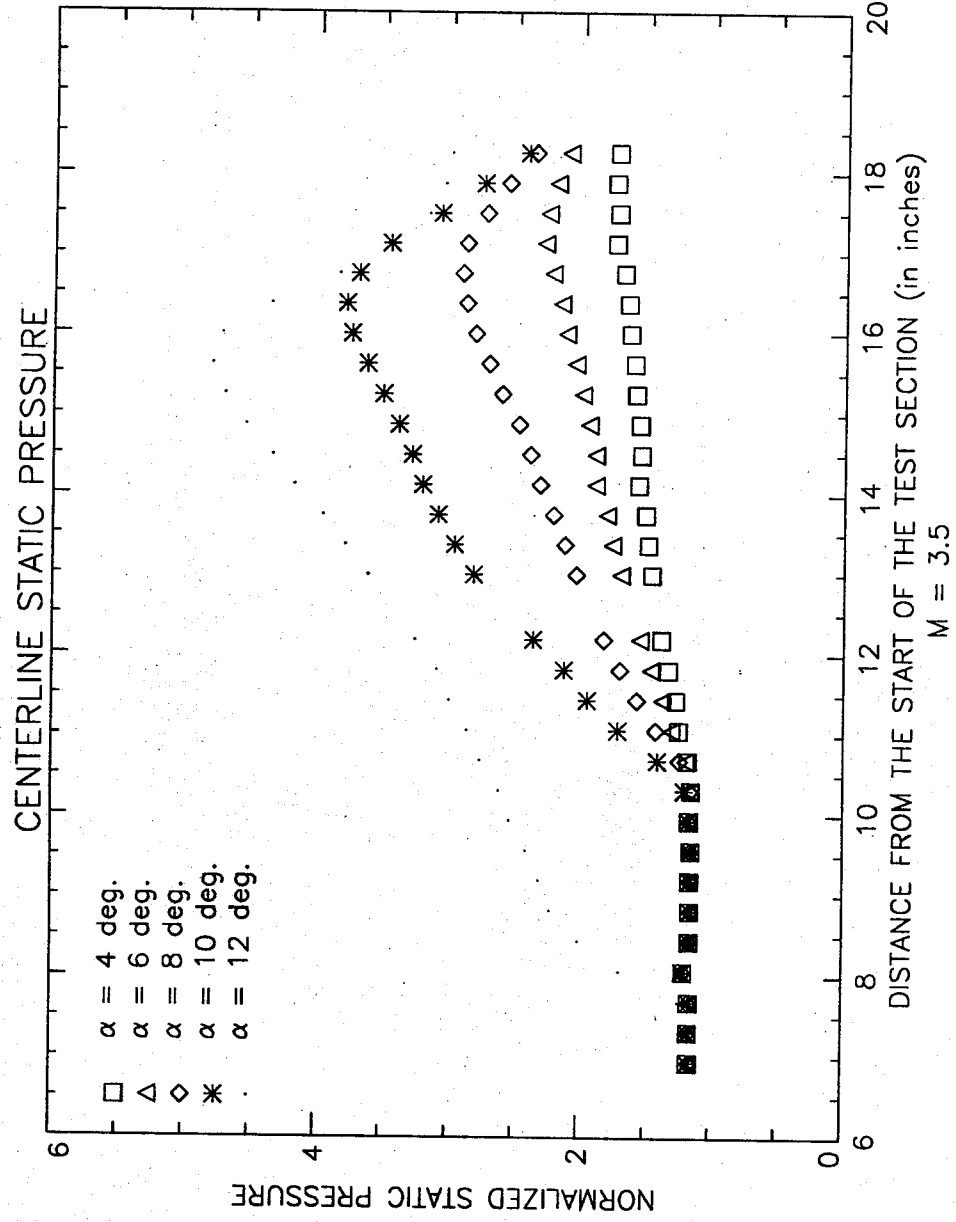








230



National Aeronautics and
Space Administration

Lewis Research Center

INTERNAL FLUID MECHANICS DIVISION



NASA LOW SPEED CENTRIFUGAL COMPRESSOR

SECOND NASA CFD VALIDATION WORKSHOP

Lewis Research Center

July 10-11, 1990

MICHAEL D. HATHAWAY

58-34
P-25
116678

198-70583

232

NASA LOW SPEED CENTRIFUGAL COMPRESSOR

- OBJECTIVES
- COMPRESSOR PROGRAM
- DESCRIPTION OF FACILITY AND COMPRESSOR
- INSTRUMENTATION
- CFD EFFORT & IMPACT ON RESEARCH PROGRAM
- CFD/EXPERIMENT PRELIMINARY COMPARISONS
- SUMMARY

234

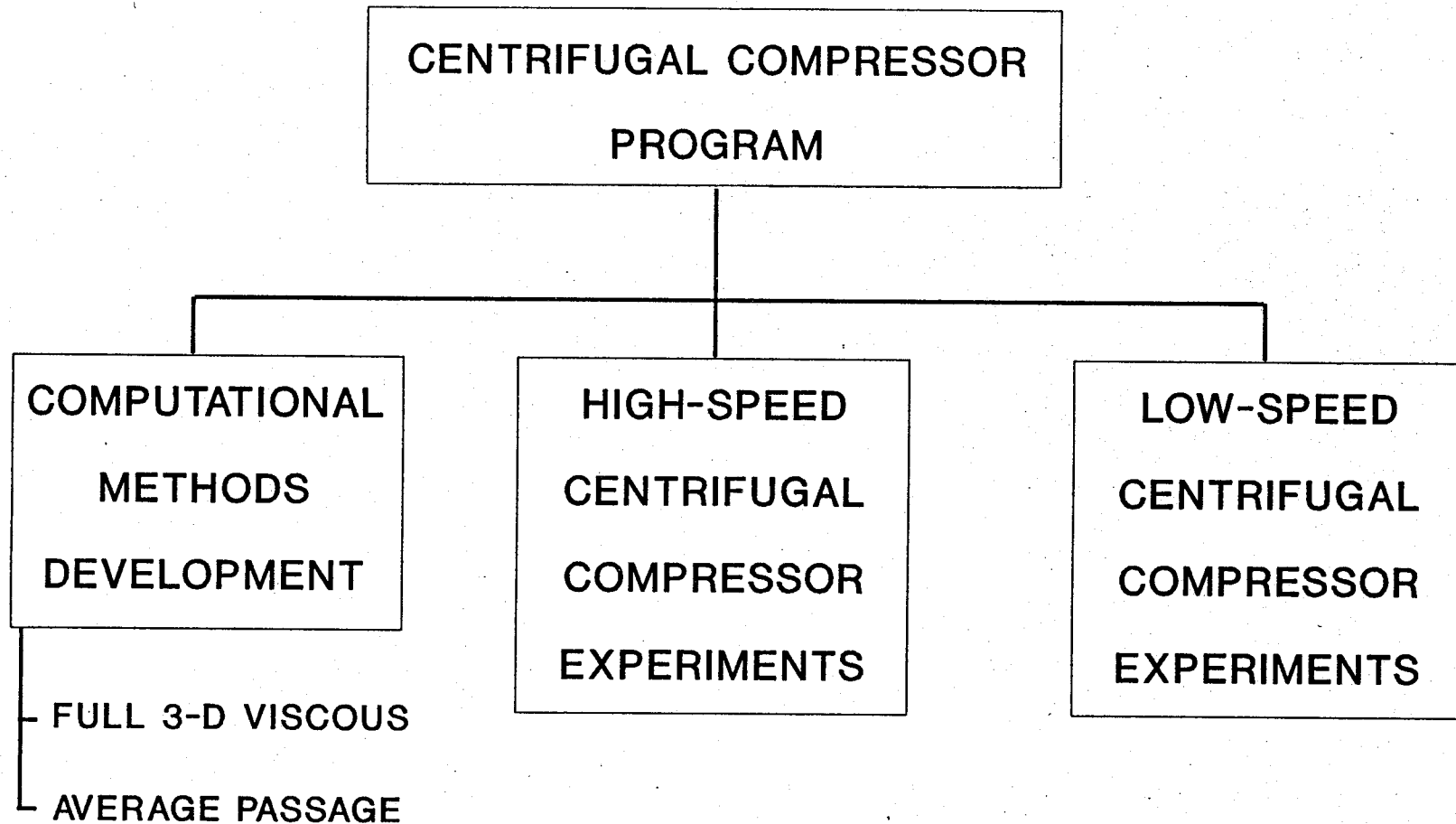
- Improve understanding of flow in centrifugal compressors
- Provide data for modeling of various flow phenomena
- Acquire "benchmark" data for 3-D viscous code validation

RESEARCH OBJECTIVES



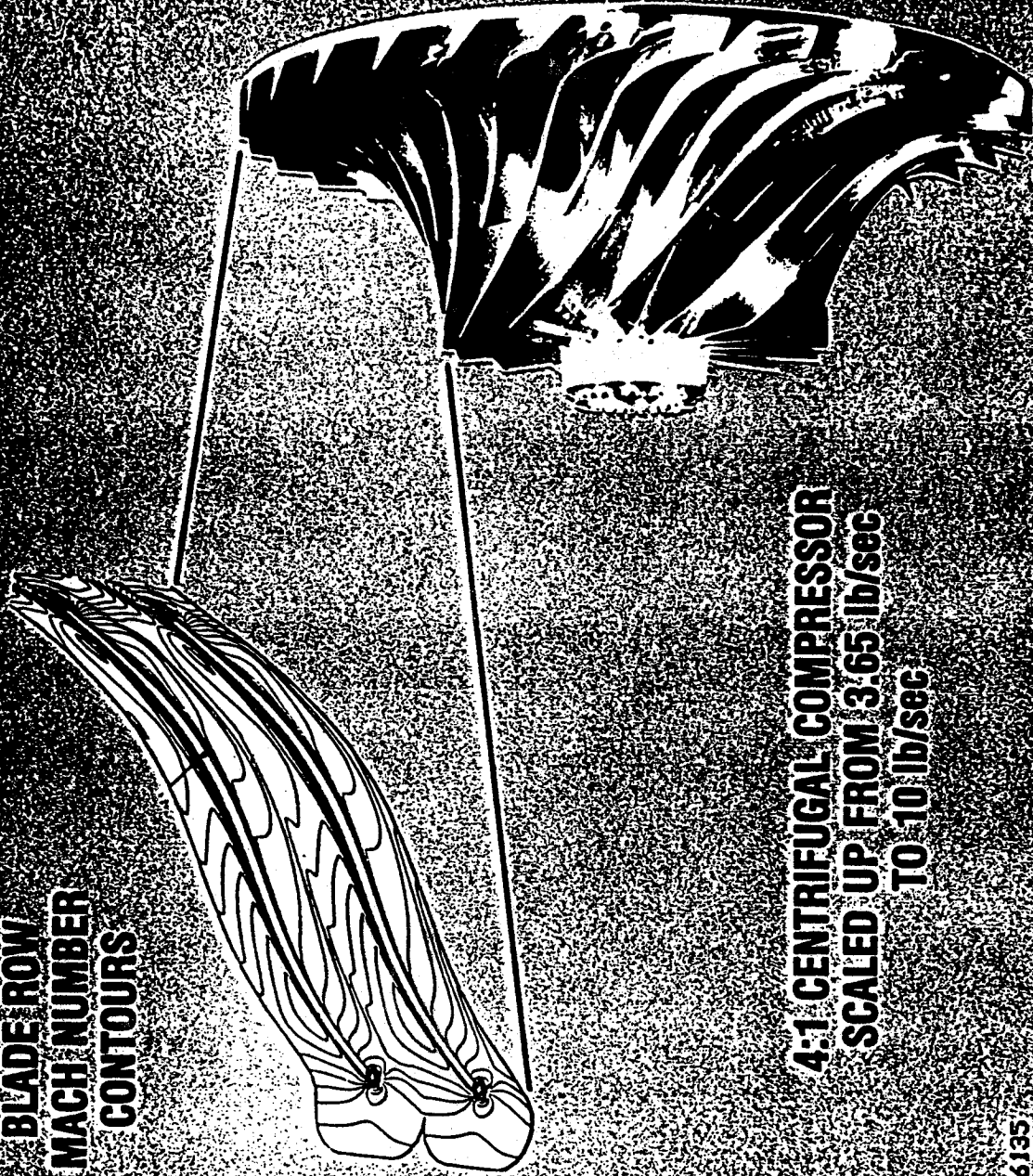
INTERNAL FLUID MECHANICS DIVISION

National Aeronautics and
Space Administration
Lewis Research Center



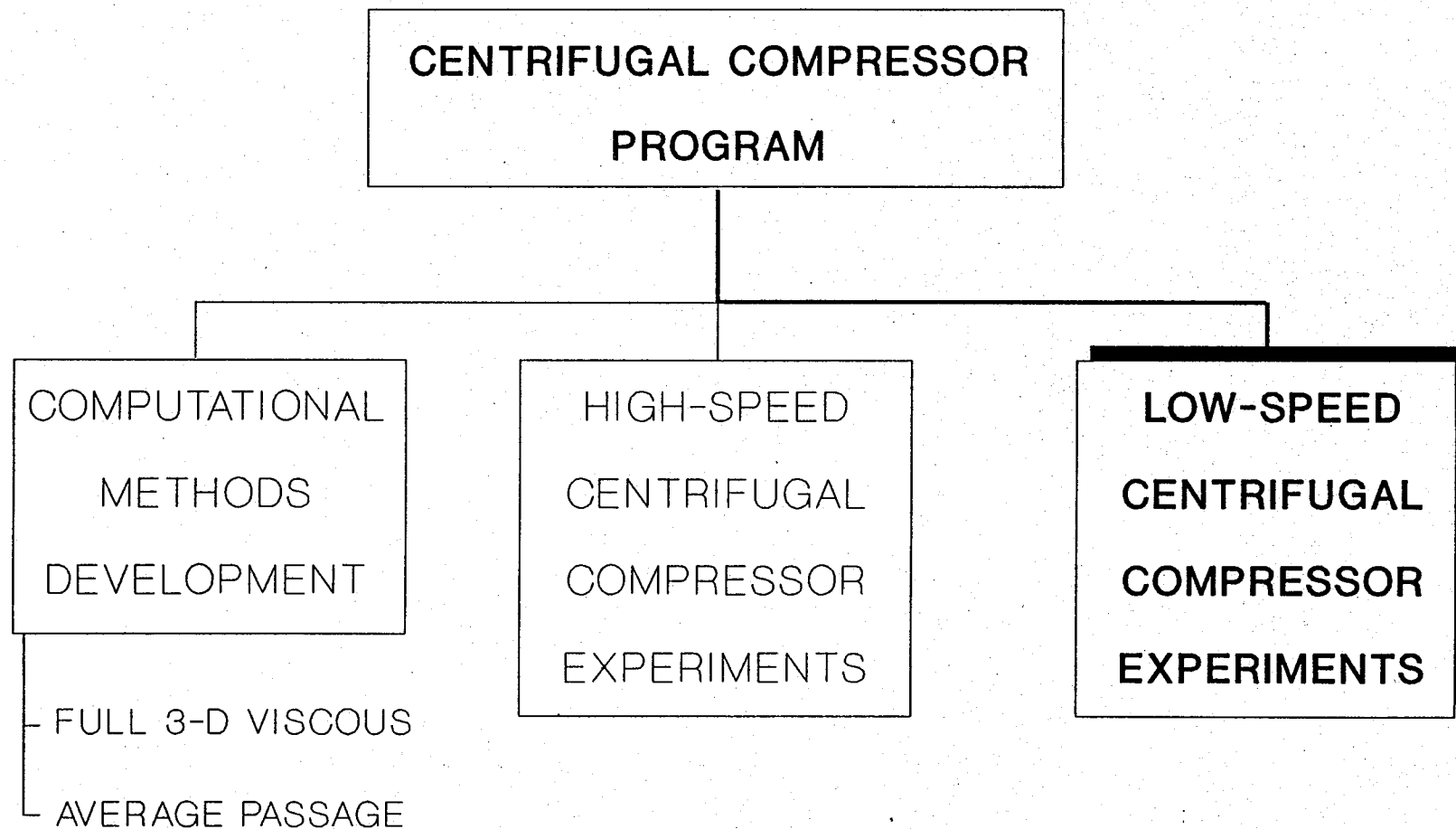
FLOW FIELD MAPPING

BLADE ROW
MACH NUMBER
CONTOURS



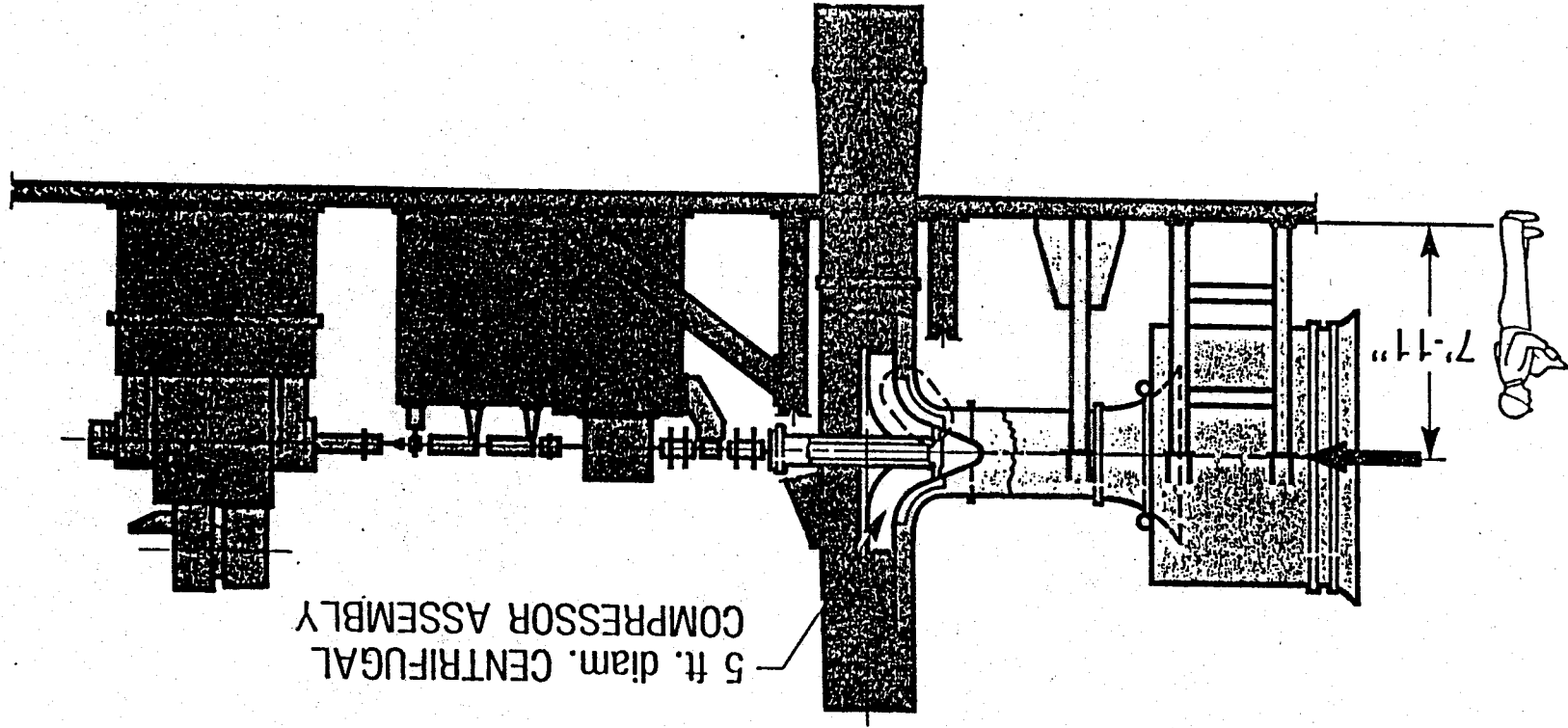
4:1 CENTRIFUGAL COMPRESSOR
SCALED UP FROM 3.65 lb/sec
TO 10 lb/sec

CD-88-36135



LOW SPEED CENTRIFUGAL COMPRESSOR BENCHMARK EXPERIMENTS FOR COMPUTATIONAL CODES

NASA
C-84-6765



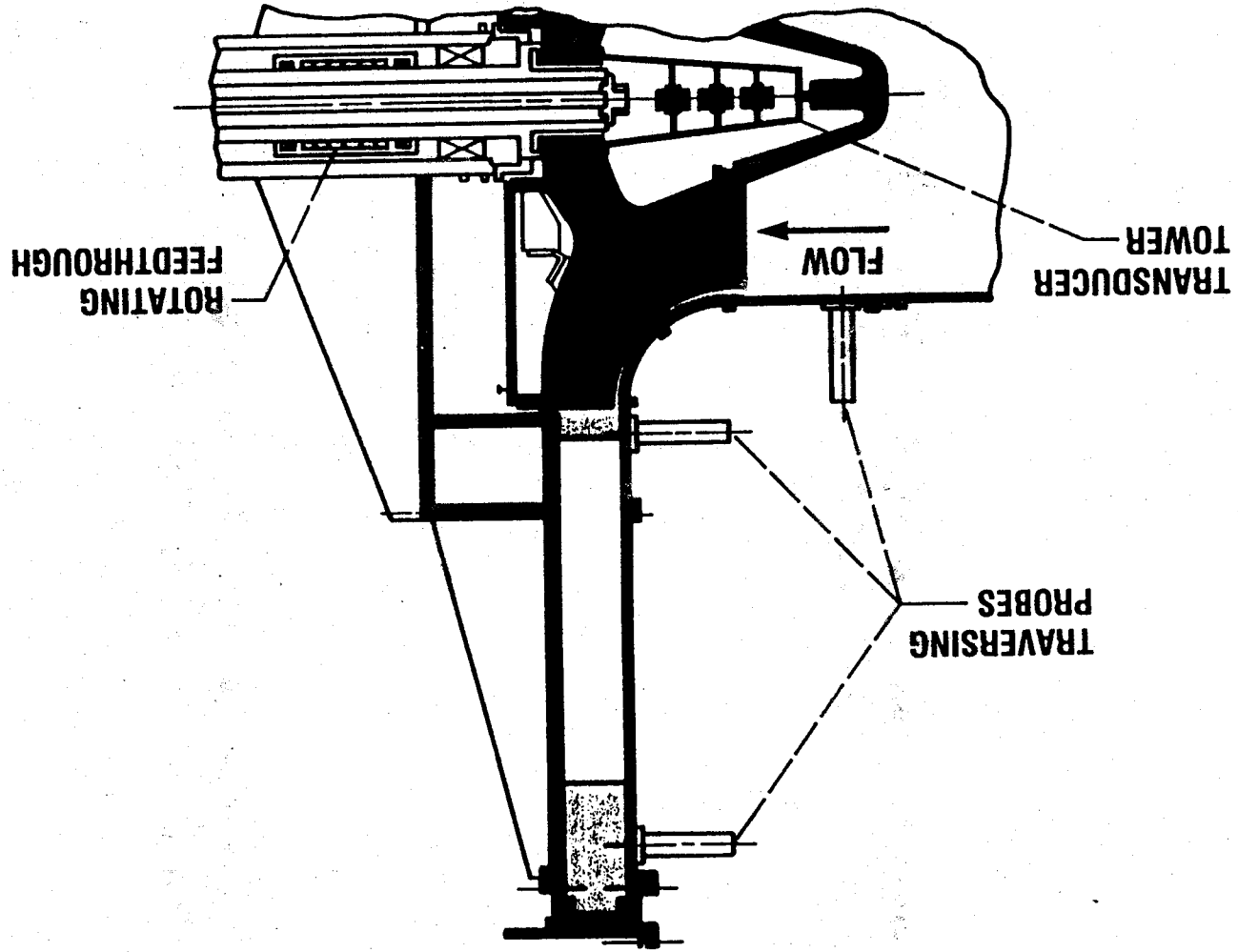
CD-84-15176

240

NASA LOW SPEED CENTRIFUGAL COMPRESSOR DESIGN CHARACTERISTICS

AIR FLOW	66 lbm/sec
ROTATIVE SPEED	1920 RPM
INLET PRESSURE	14.7 psia
PRESSURE RATIO	1.18
OUTLET TEMP.	85 F
EXHAUST	ALTITUDE

LOW SPEED CENTRIFUGAL COMPRESSOR COMPRESSOR INSTRUMENTATION

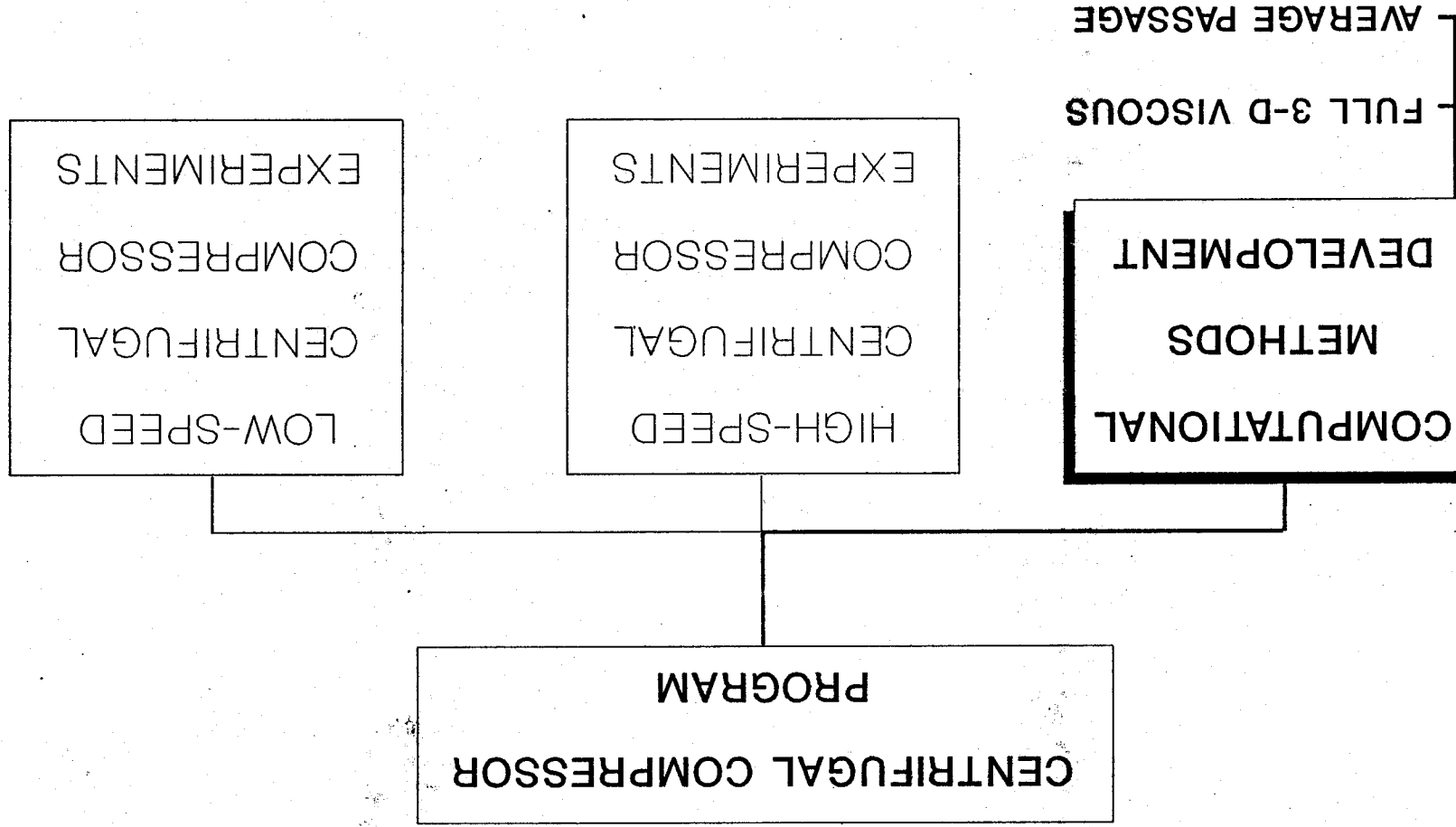


CD-90-45124

h7c

NASA LOW SPEED CENTRIFUGAL COMPRESSOR INSTRUMENTATION

- 465 STATIC PRESSURES (208 ON ROTOR)
- 83 TOTAL PRESSURES (75 ON ROTOR)
- 51 TOTAL TEMPERATURES (27 ON ROTOR)
- 8 FLOW ANGLE PROBES
- 28 STRAIN GAUGES
- 6 PROXIMITY PROBES
- LASER ANEMOMETRY



LOW SPEED COMPRESSOR RESEARCH PROGRAM

CENTRIFUGAL COMPRESSOR - CFD APPLICATIONS

OBJECTIVE:

Perform CFD analysis in order to provide guidance for experimental measurements:

- develop a "feel" for the flow physics
- aid in planning location/extent of measurements
- on-line assessment of CFD limitations

APPROACH:

3D steady Navier-Stokes analysis of impeller only

- VPI&SU grant
- in-house

3D steady Average-Passage analysis of impeller+diffuser

STATUS:

VPI&SU impeller analysis complete

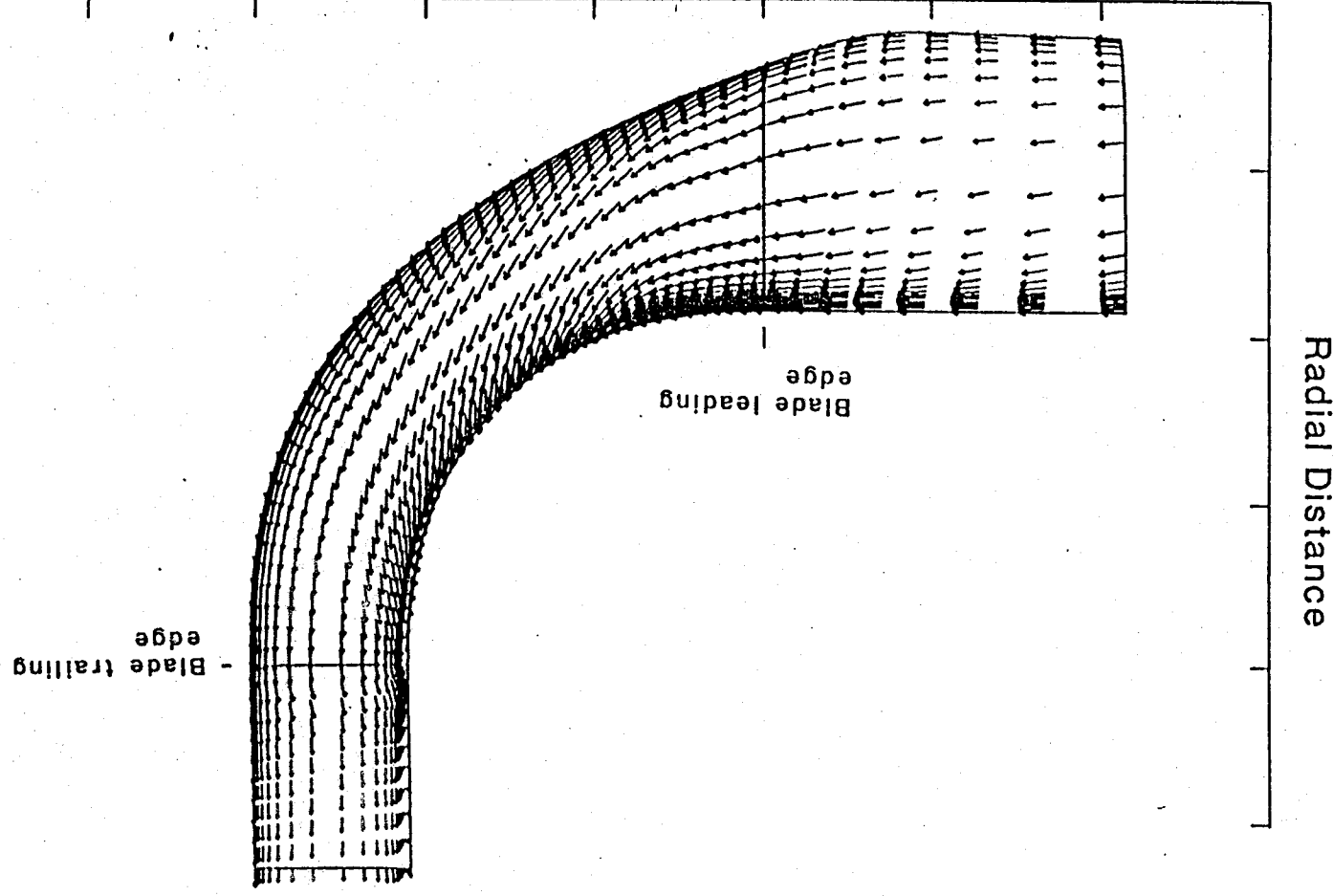
In-house impeller analysis in progress

Average-passage stage analysis in progress

LSCR-3

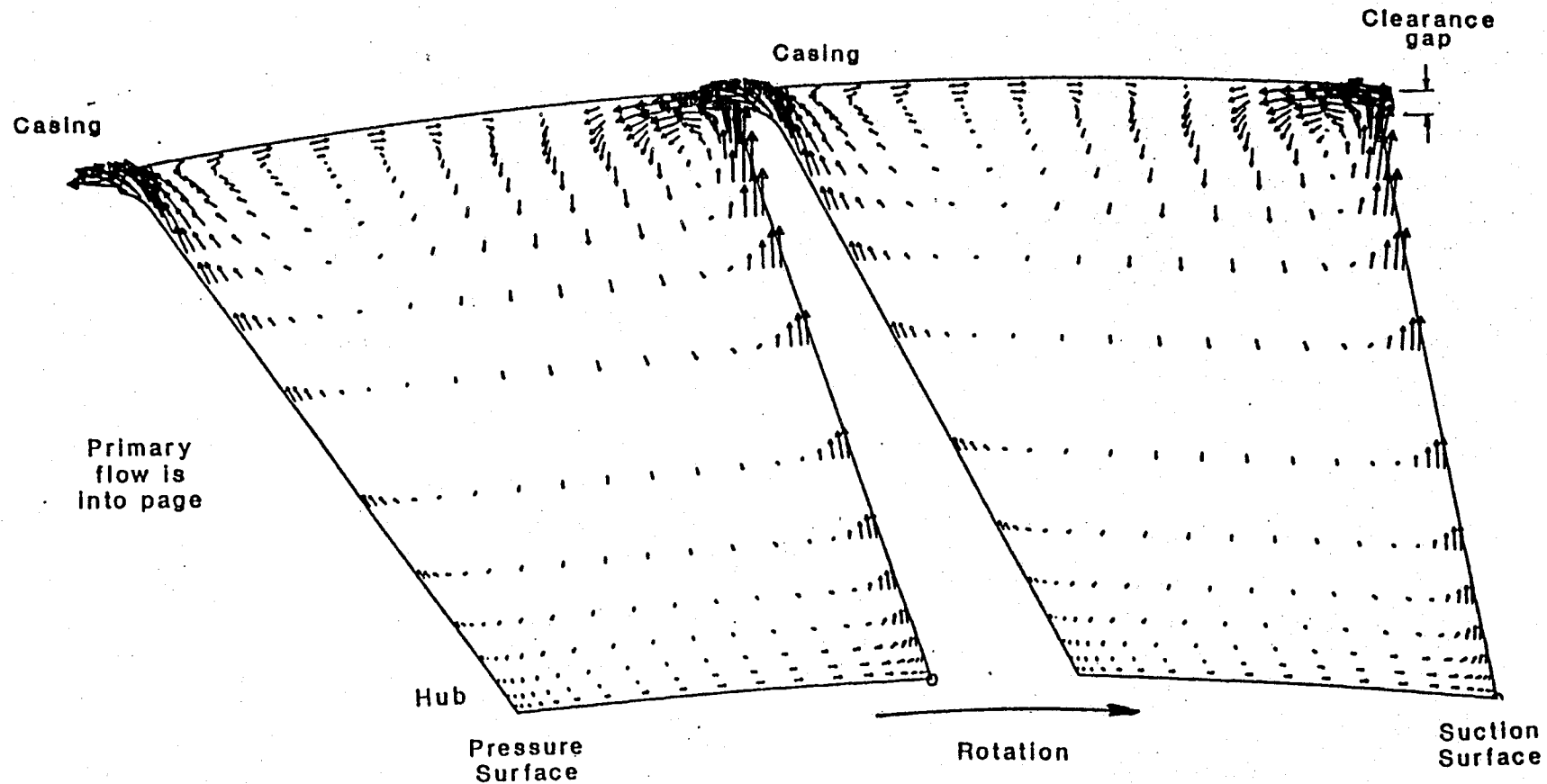
3D NAVIER STOKES CODE ANALYSIS OF LARGE LOW-SPEED CENTRIFUGAL IMPELLER

PRESSURE DRIVEN SECONDARY FLOWS
NEAR BLADE SUCTION SURFACE

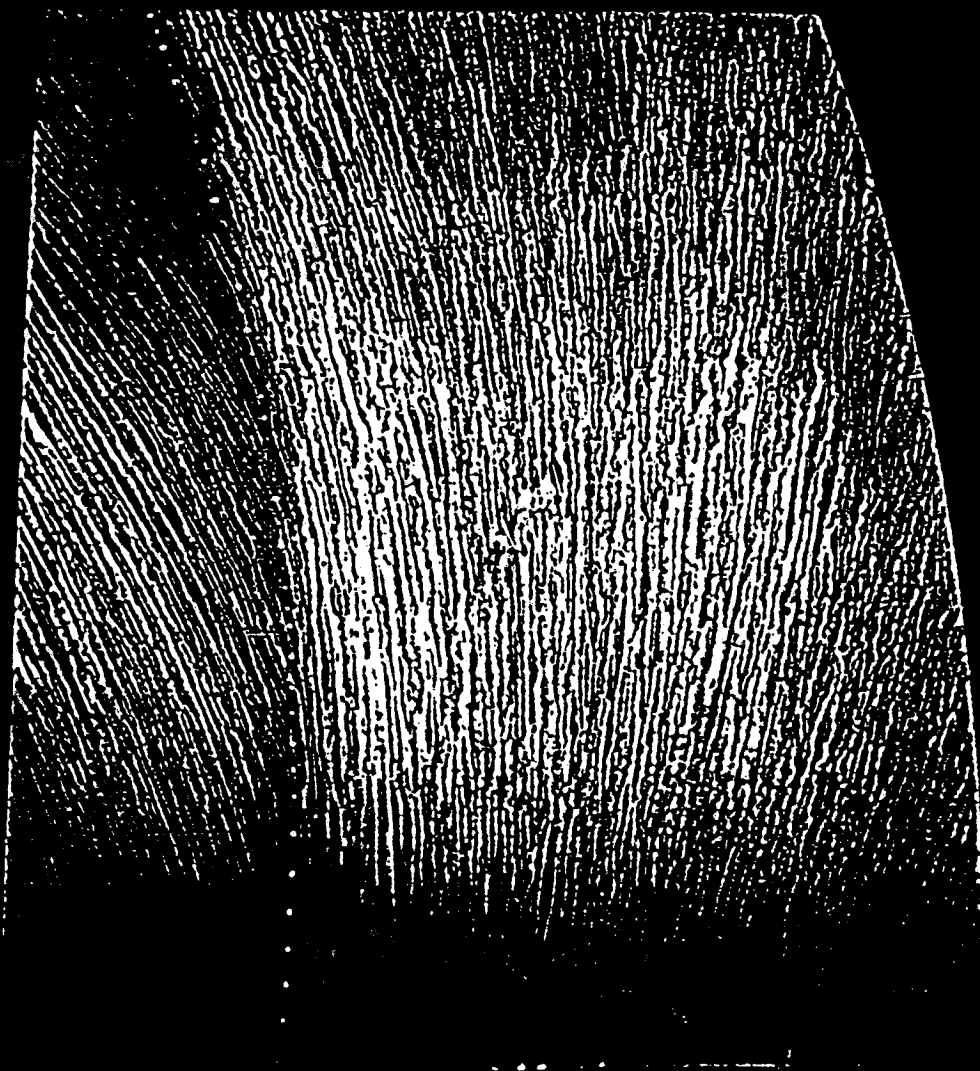


3D NAVIER STOKES CODE ANALYSIS OF LARGE LOW-SPEED CENTRIFUGAL IMPELLER

PRESSURE DRIVEN
TIP CLEARANCE FLOW



NASA
C-90-05975



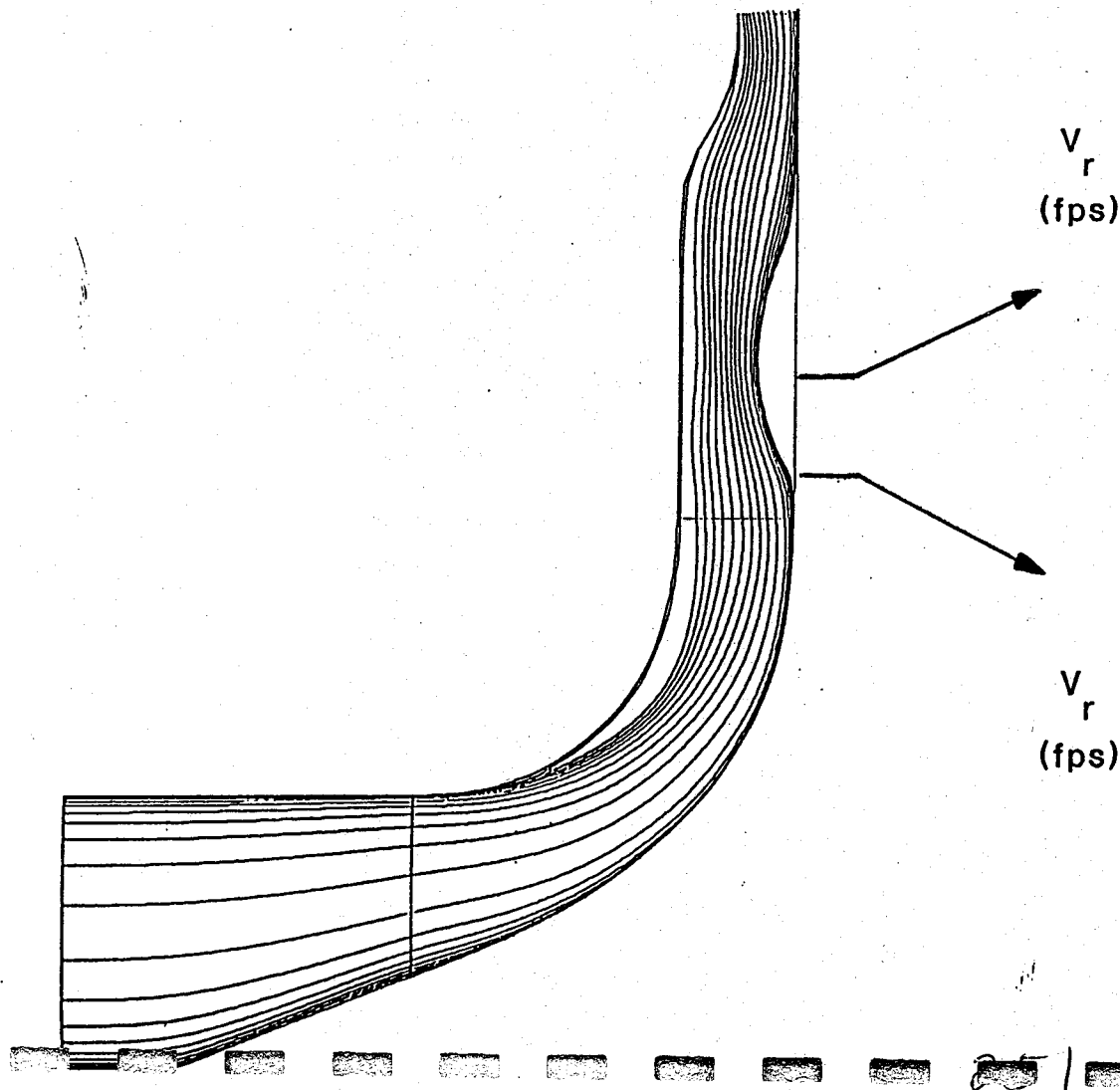
LOW SPEED CENTRIFUGAL COMPRESSOR
DIFFUSER SHROUD
66 lbm/sec

LOW SPEED CENTRIFUGAL COMPRESSOR

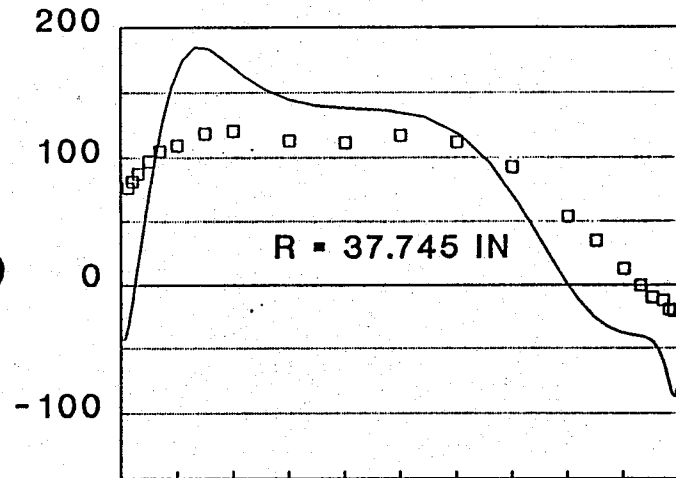
3-D NS CODE PREDICTION V.S. EXPERIMENT

3-D NAVIER STOKES ANALYSIS
PARTICLE TRACES

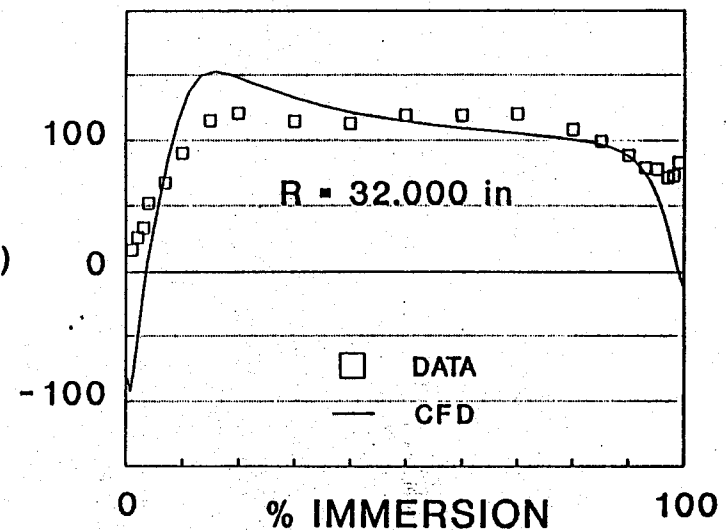
IMPELLER EXIT
RADIAL VELOCITY DISTRIBUTION



V_r
(fps)



V_r
(fps)



LOW SPEED CENTRIFUGAL COMPRESSOR
DIFFUSER HUB
66 lbm/sec



LeRC COMPRESSOR FLOW PHYSICS EXPERIMENT SCHEDULE

	90	91	92	93	94	95
--	----	----	----	----	----	----

Low Speed Compressor Facility - IFMD

Centrifugal

rotor
only

full stage

Multistage Axial

4-stage axial
compressor

Small High Speed Compressor Facility - PSD

centrifugal
compressor

3-stage axial
compressor

LSA-363

SUMMARY

- **LOW SPEED CENTRIFUGAL COMPRESSOR FACILITY**
 - ✓ Improve understanding of complex flows
 - ✓ Data for flow physics modeling
 - ✓ "Benchmark" data for code assessment
- **CFD ANALYSIS**
 - ✓ Develop feel for flow physics
 - ✓ Aid in planning location/extent of measurements
 - ✓ On-line assessment of CFD limitations
- **HARDWARE MODIFICATIONS**
 - ✓ 2% Tip clearance
 - ✓ Diffuser wedge plate
- **MILESTONES**
 - ✓ Laser anemometer surveys LSCC, mid Sept.
 - ✓ High speed centrifugal, LFA surveys, 1990
 - ✓ Low speed axial compressor, May 1991
 - ✓ Impeller/Diffuser LSCC, 1993



National Aeronautics and
Space Administration

July 10-12, 1990

George C. Marshall Space Flight Center
Marshall Space Flight Center, Alabama 35812

MSFC CFD VALIDATIONS - CURVED DUCT AND TURBOMACHINERY FLOWFIELD ANALYSES

Omit
to
End
~~7-35~~

SECOND NASA CFD VALIDATION WORKSHOP NASA LEWIS RESEARCH CENTER

PREPARED BY: LISA W. GRIFFIN , ROBERT WILLIAMS, AND PAUL MCCONNAUGHEY
COMPUTATIONAL FLUID DYNAMICS BRANCH
AEROPHYSICS DIVISION
STRUCTURES AND DYNAMICS LABORATORY

OVERVIEW

- DUCT AND PIPE BEND FLOW CODE VALIDATION
- 90° DUCT BEND
 - OBJECTIVES
 - APPLICATION CONFIGURATION
 - CODE METHODOLOGY
 - RESULTS
 - COMPUTER RESOURCE REQUIREMENTS
 - CONCLUSIONS
- TURBOMACHINERY CODE VALIDATION
- TURBINE CASCADES
 - OBJECTIVES
 - APPROACH
 - APPLICATIONS CONFIGURATIONS
 - CODE METHODOLOGY
 - RESULTS
 - COMPUTER RESOURCE REQUIREMENTS
 - CONCLUSIONS

DUCT AND PIPE BEND FLOW CODE VALIDATION

<u>CODE</u>	<u>APPLICATION</u>	<u>PRINCIPAL INVESTIGATORS</u>
INS3D	* 90° DUCT BEND S-SHAPED DUCT BEND 180° TURNAROUND DUCT	PAUL MCCONNAUGHEY, JONI CORNELISON
INS3DLU	* 90° DUCT BEND 90° PIPE BEND	ROBERT WILLIAMS
FDNS3D	* 90° DUCT BEND 90° PIPE BEND	ROBERT WILLIAMS

OBJECTIVE

**ASSESS CODE CAPABILITY TO PREDICT COMPLEX SECONDARY
FLOWS AND ASSOCIATED AXIAL FLOW VARIATION**

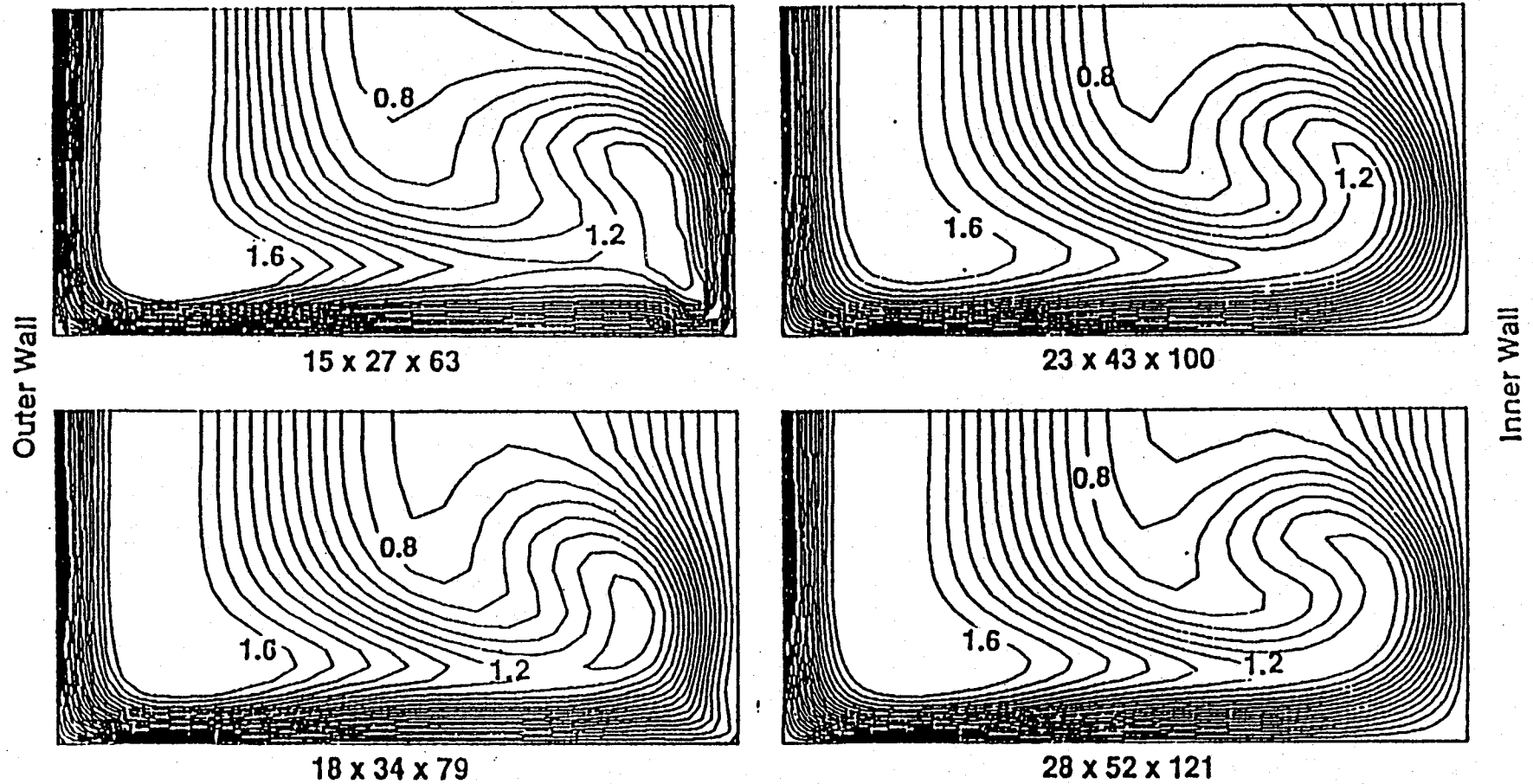
APPLICATION CONFIGURATION

- **NINETY DEGREE DUCT BEND WITH CONSTANT, SQUARE CROSS SECTION (TAYLOR, WHITELAW, AND YANNESKIS)**
 - **RADIUS RATIO - 2.3**
 - **REYNOLDS NUMBER - 790**
 - **DEAN NUMBER - 368**
- **LASER-DOPPLER VELOCIMETER DATA**
 - **STREAMWISE AND GAPWISE VELOCITY COMPONENTS**
 - **8 STREAMWISE STATIONS**
 - **50 MEASUREMENTS PER STATION**

CODE METHODOLOGY

- **INS3D**
 - **INCOMPRESSIBLE NAVIER-STOKES CODE**
 - **BEAM-WARMING /BRILEY MCDONALD ALGORITHM**
 - **ARTIFICIAL COMPRESSIBILITY**
 - **2ND OR 4TH ORDER NUMERICAL DISSIPATION MODEL**
- **INS3DLU**
 - **INCOMPRESSIBLE NAVIER-STOKES CODE**
 - **LOWER-UPPER SYMMETRIC-GAUSS-SIDEL IMPLICIT SCHEME**
 - **ARTIFICIAL COMPRESSIBILITY**
 - **3RD ORDER NUMERICAL DISSIPATION MODEL**
- **FDNS3D**
 - **UNSTEADY REYNOLDS-AVERAGED NAVIER-STOKES CODE**
 - **PRESSURE-BASED PREDICTOR-CORRECTOR PISO TYPE ALGORITHM**
 - **VARIABLE ORDER UPWIND DISSIPATION**
 - **K- ϵ TURBULENCE MODEL FOR TURBULENT FLOWS**
 - **EQUILIBRIUM/FINITE RATE CHEMISTRY FOR REACTIVE FLOWS**

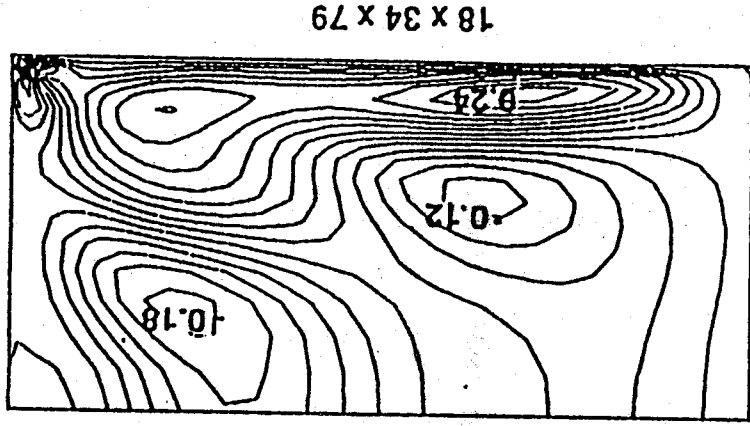
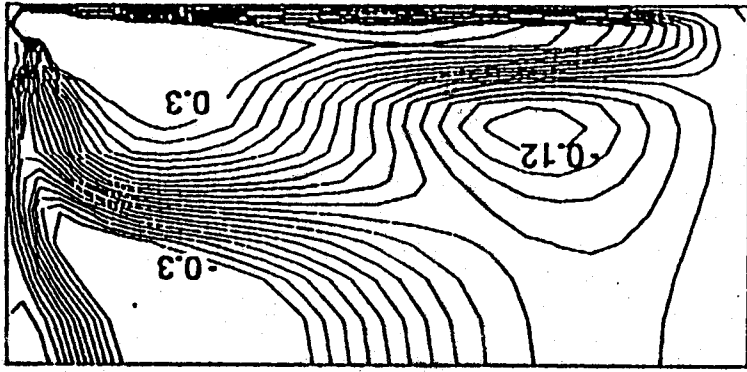
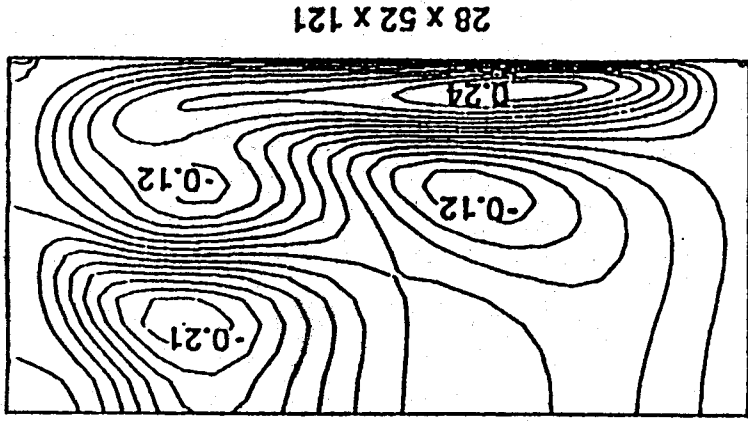
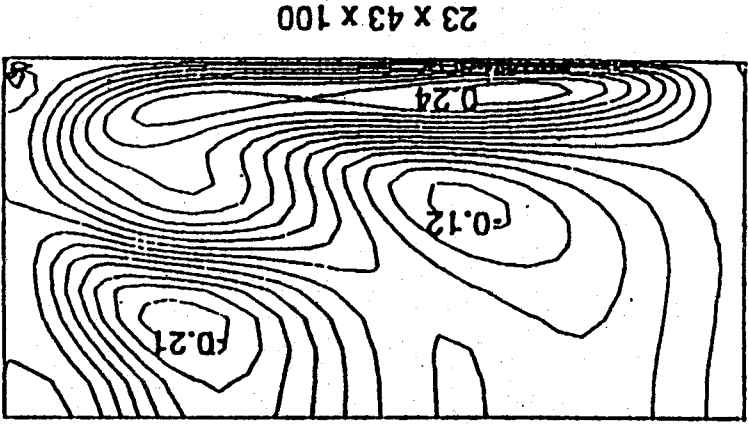
Streamwise Velocity Contours at the Plane 0.25 H Past the 90° Bend Exit.



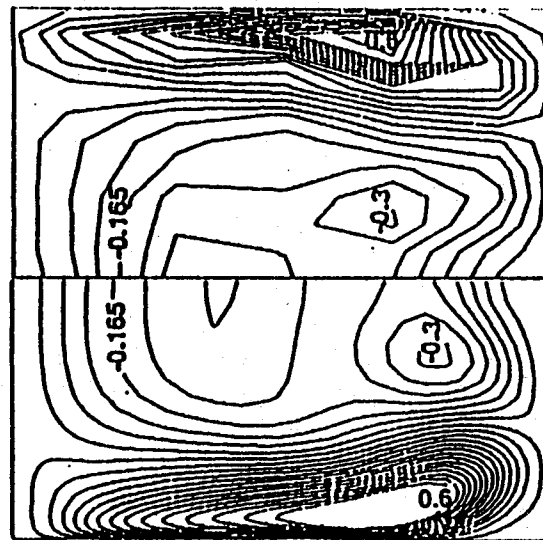
Gapwise Velocity Contours at the Plane 0.25 H Past the 90° Bend Exit.

Inner Wall

Outer Wall

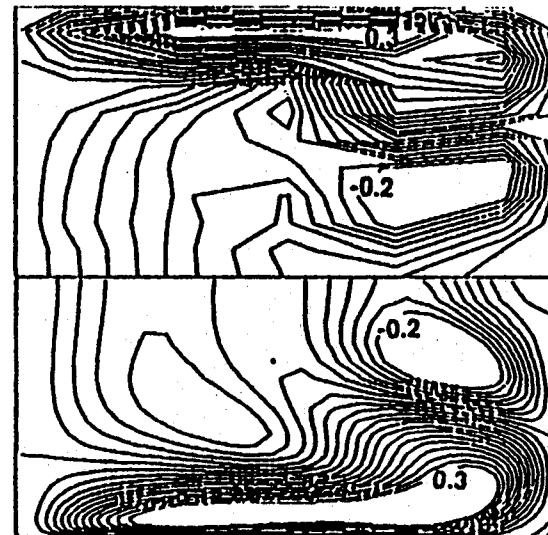


Outer Wall



$\theta = 60^\circ$
Cont. Int. = 0.045

Exp.

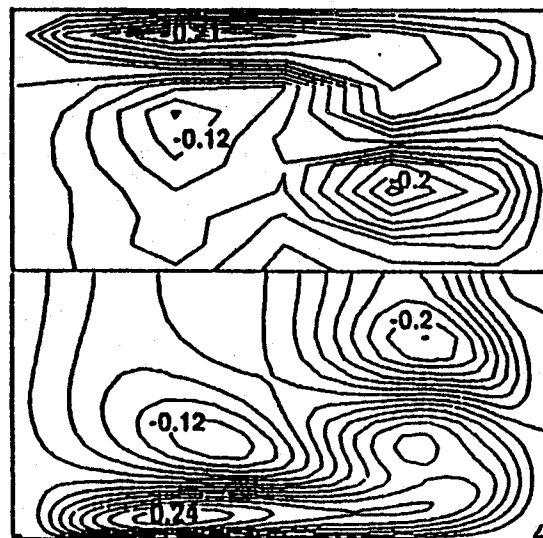


Inner Wall

Comp.

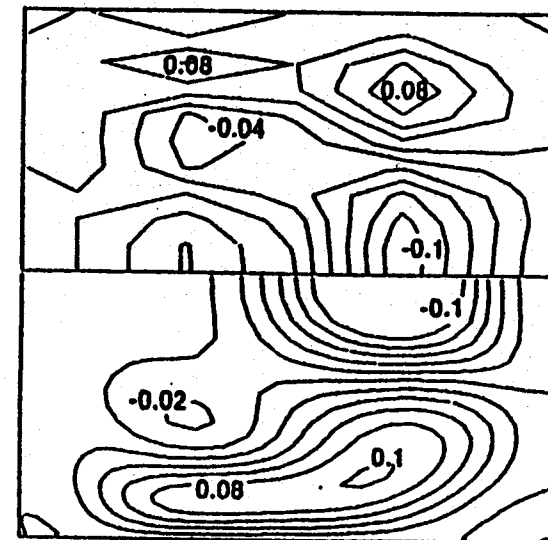
$\theta = 77.5^\circ$
Cont Int. = 0.025

Outer Wall



$X = +0.25 H$
Cont. Int. = 0.03

Exp.



Inner Wall

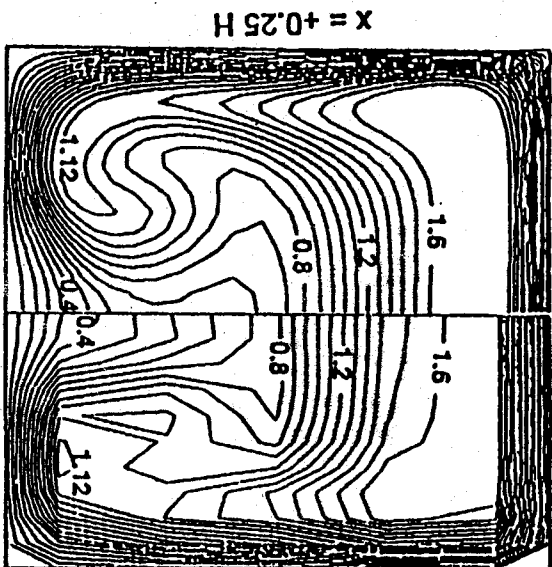
Comp.

$X = +2.5 H$
Cont Int. = 0.02

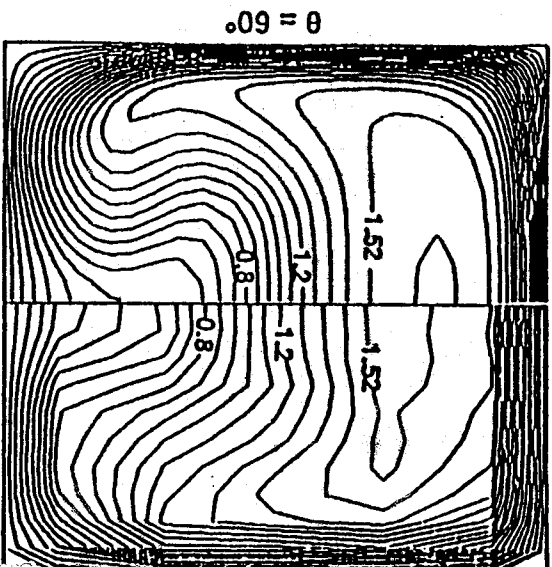
Predicted Radial Flow In 90° Bend Compared with the Data of Taylor et al.⁹

Predicted Axial Flow in 90° Bend Compared with the Data of Taylor et al.
Contour Interval of Plots is 0.08.

Outer Wall



Outer Wall



Comp.

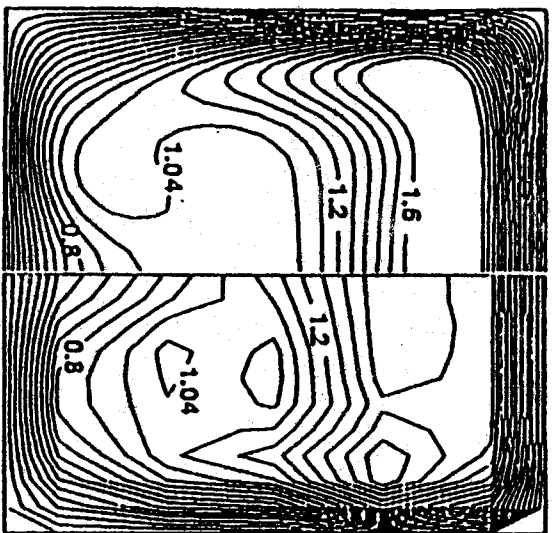
Exp.

Comp.

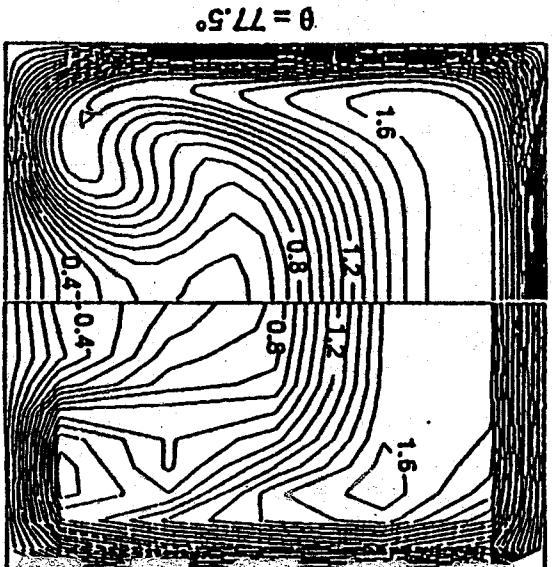
Exp.

$x = 2.5 H$

Inner Wall



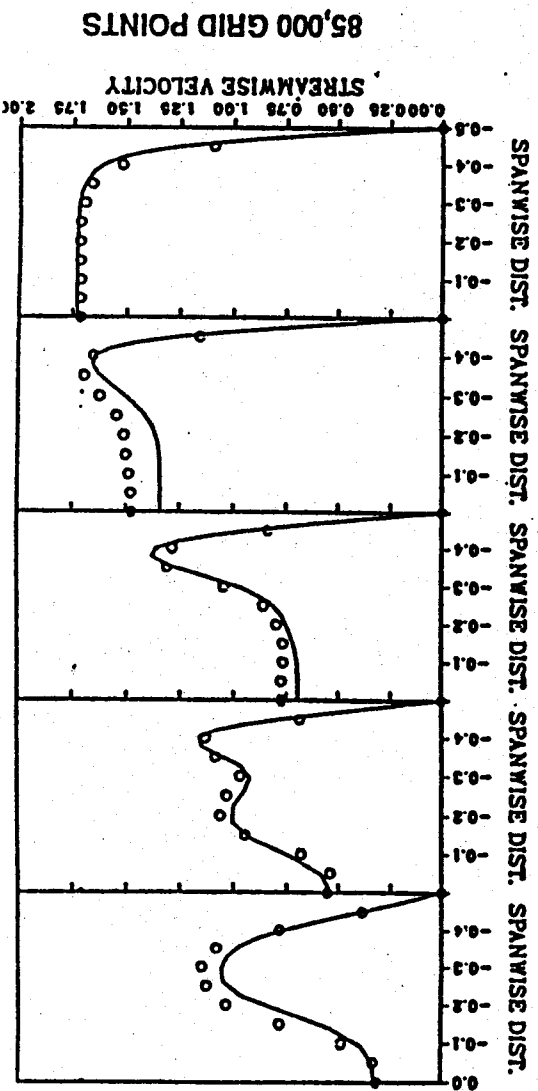
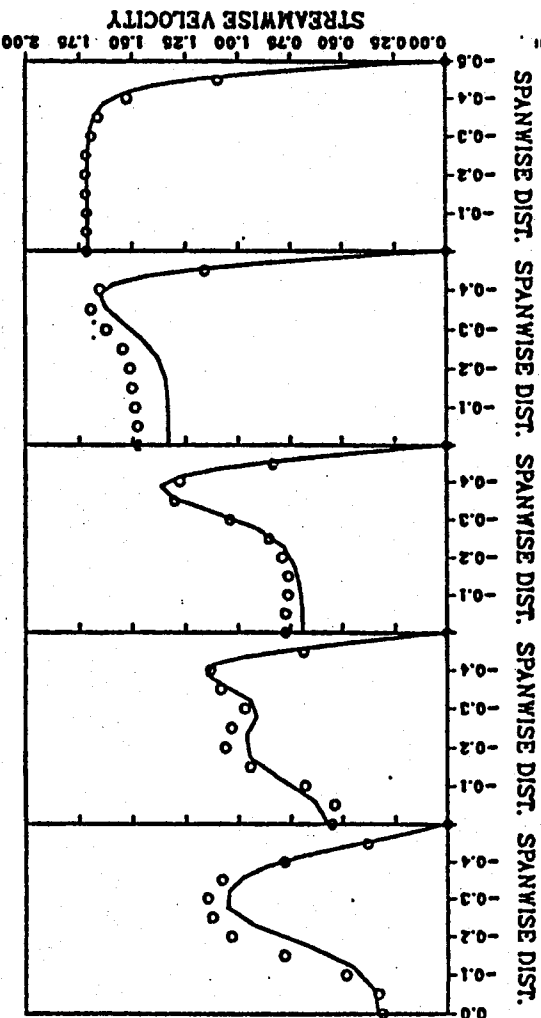
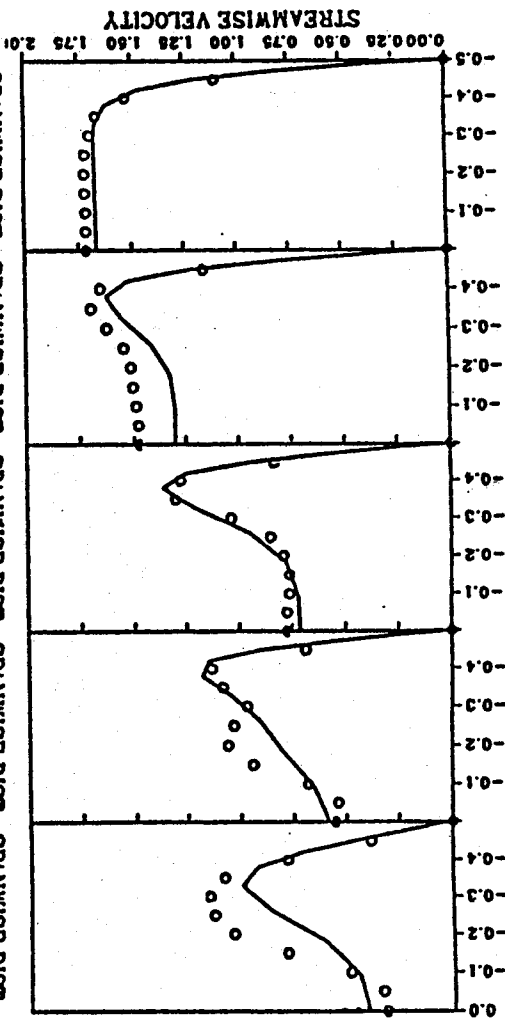
Inner Wall



STREAMWISE VELOCITY PROFILES - INS3DLU

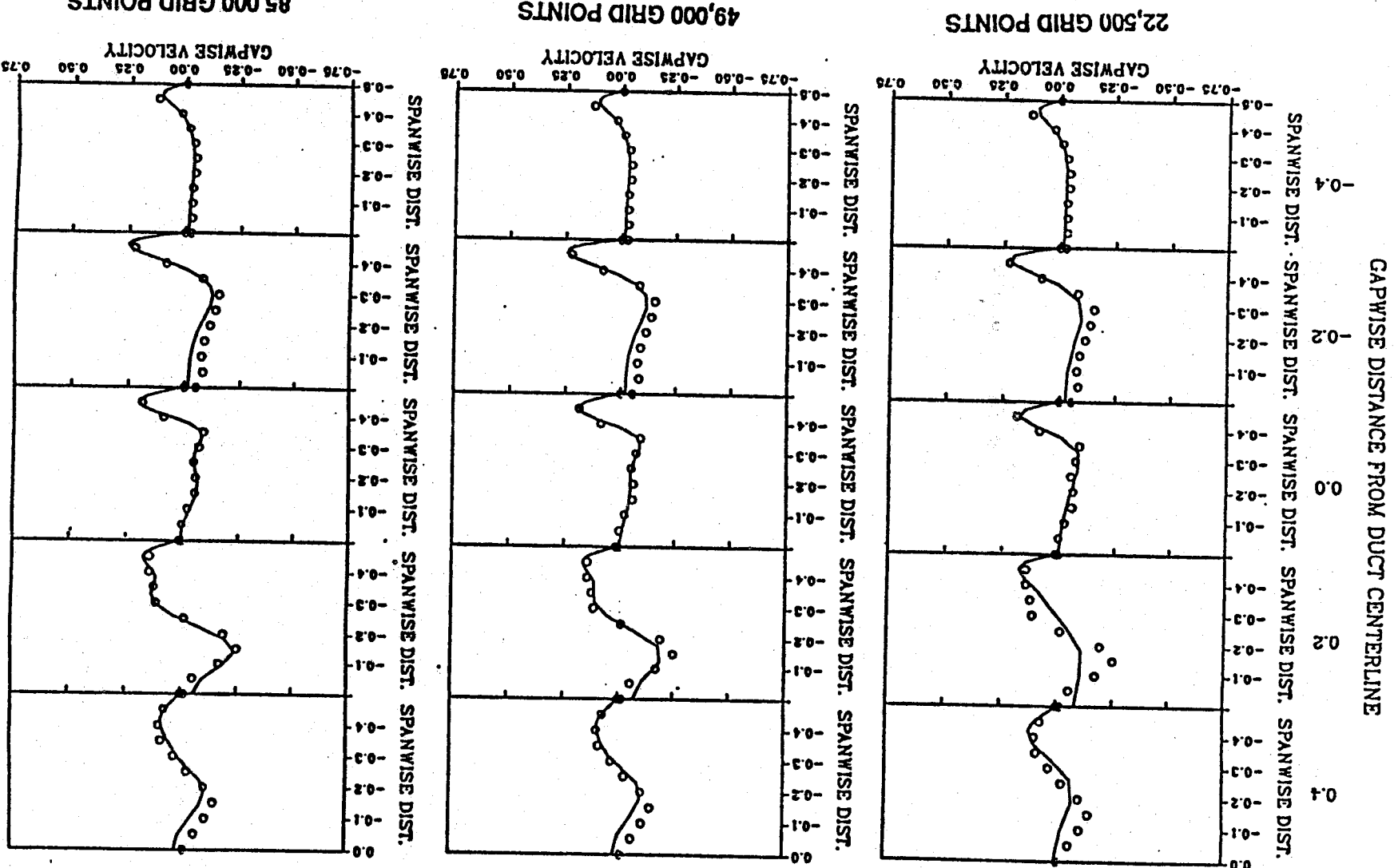
CAPWISE DISTANCE FROM DUCT CENTERLINE

SPANWISE DIST. SPANWISE DIST. SPANWISE DIST. SPANWISE DIST. SPANWISE DIST.



0.25 H Past the 90° Bend Exit

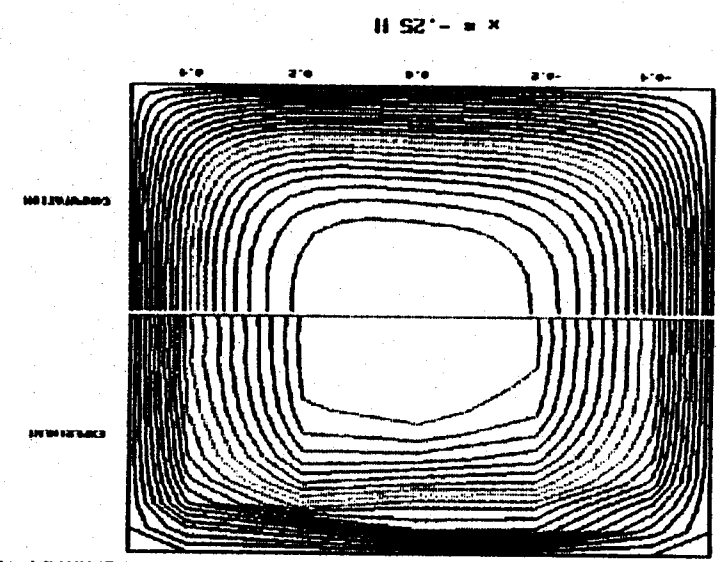
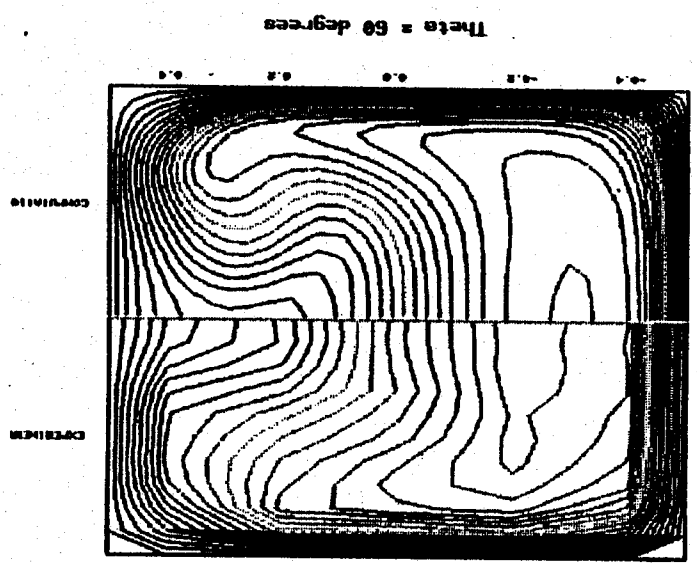
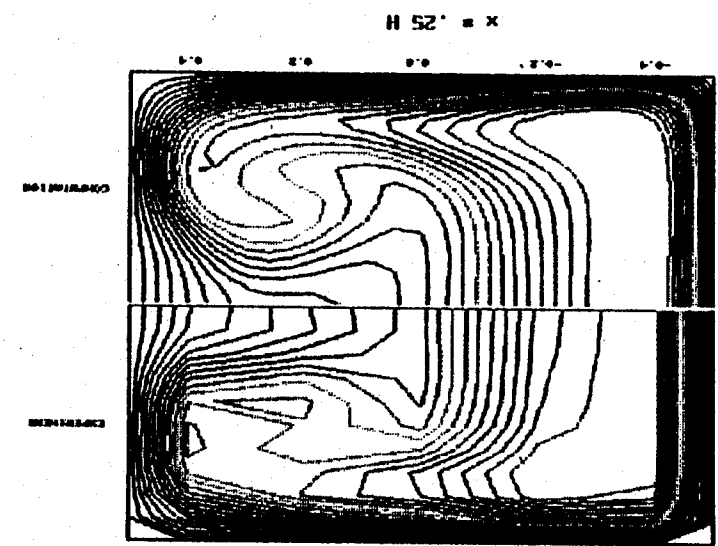
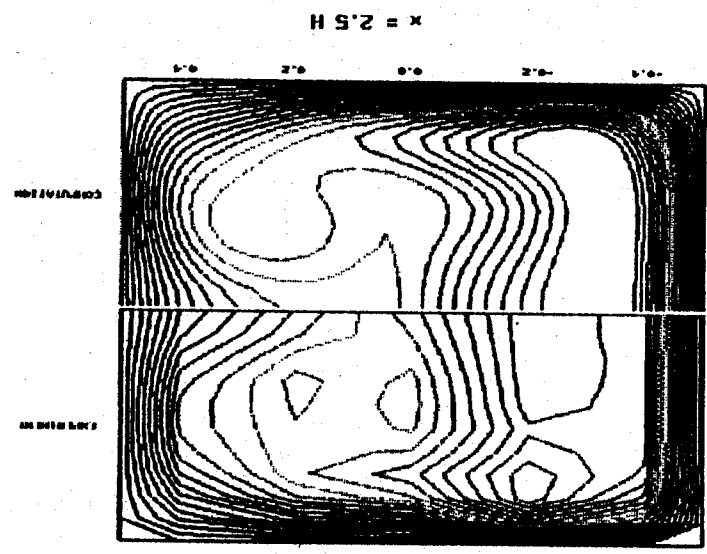
GAPWISE VELOCITY PROFILES - INS3DLU



0.25 H Past the 90° Bend Exit

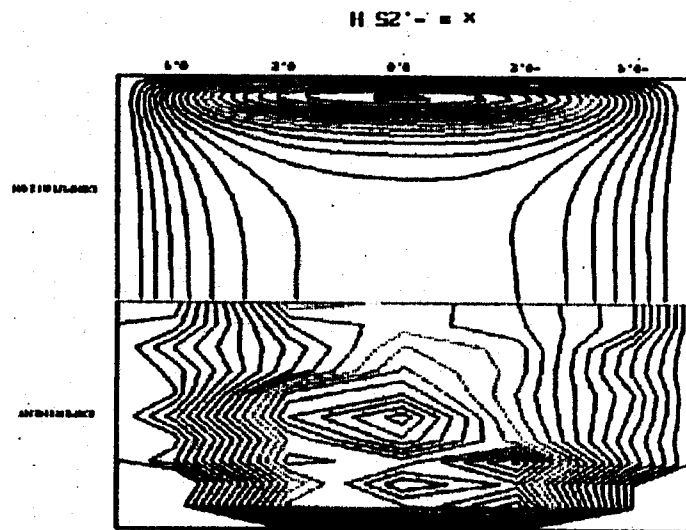
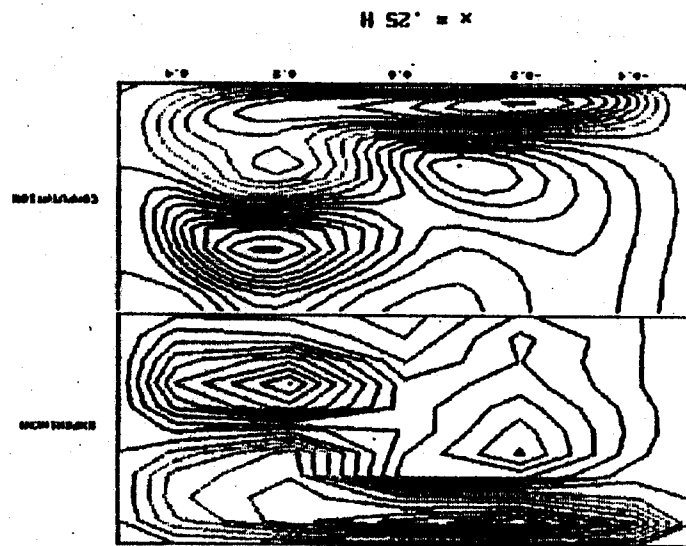
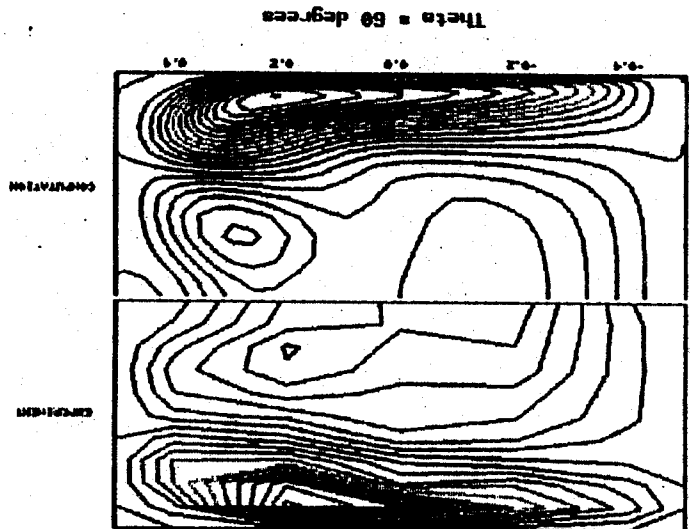
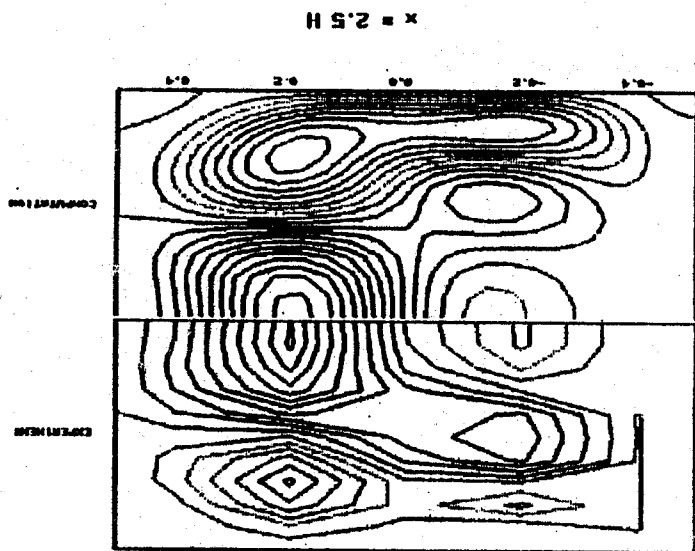
268

1-20

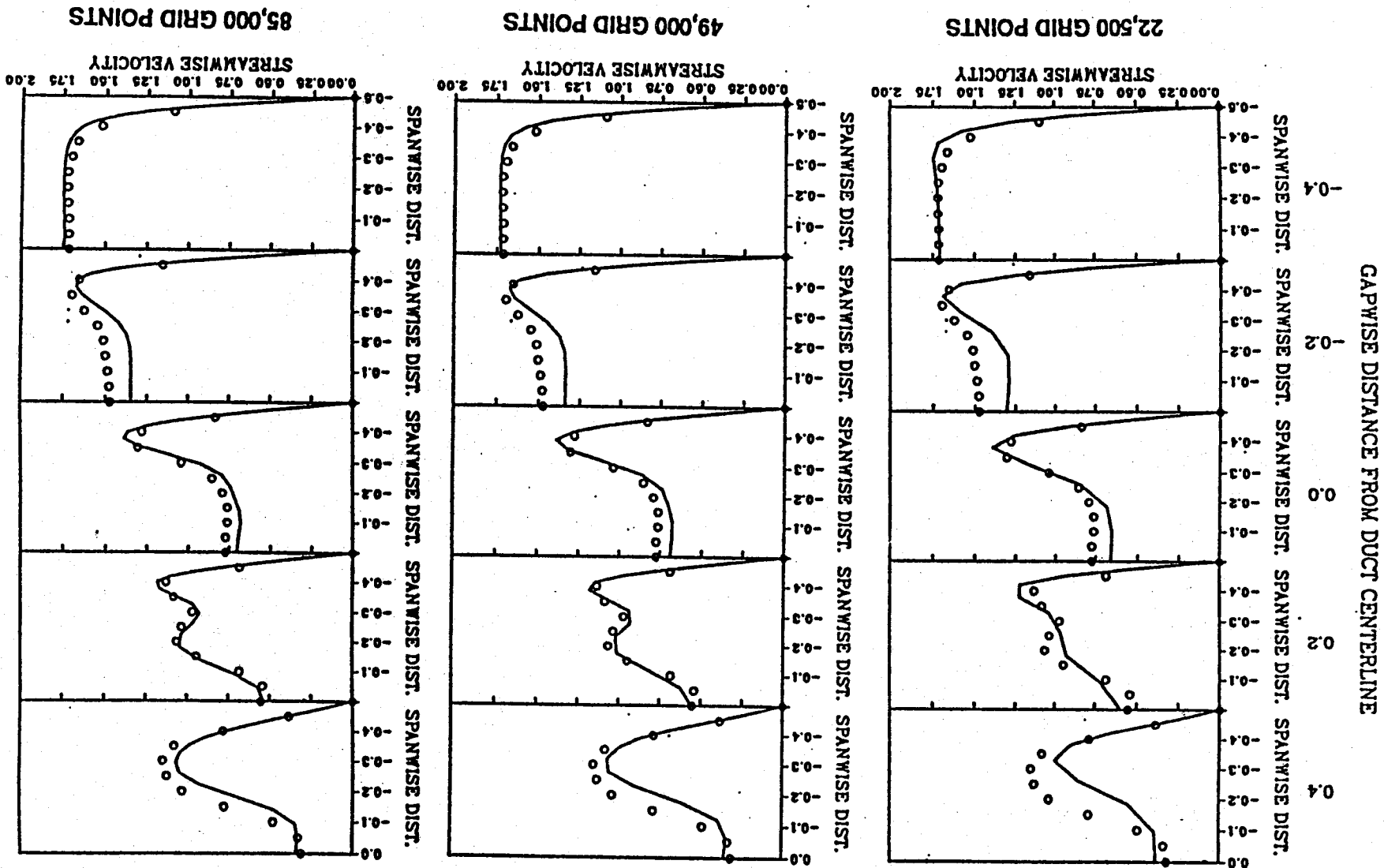


Streamlines: Velocity Contours - INS3DLU

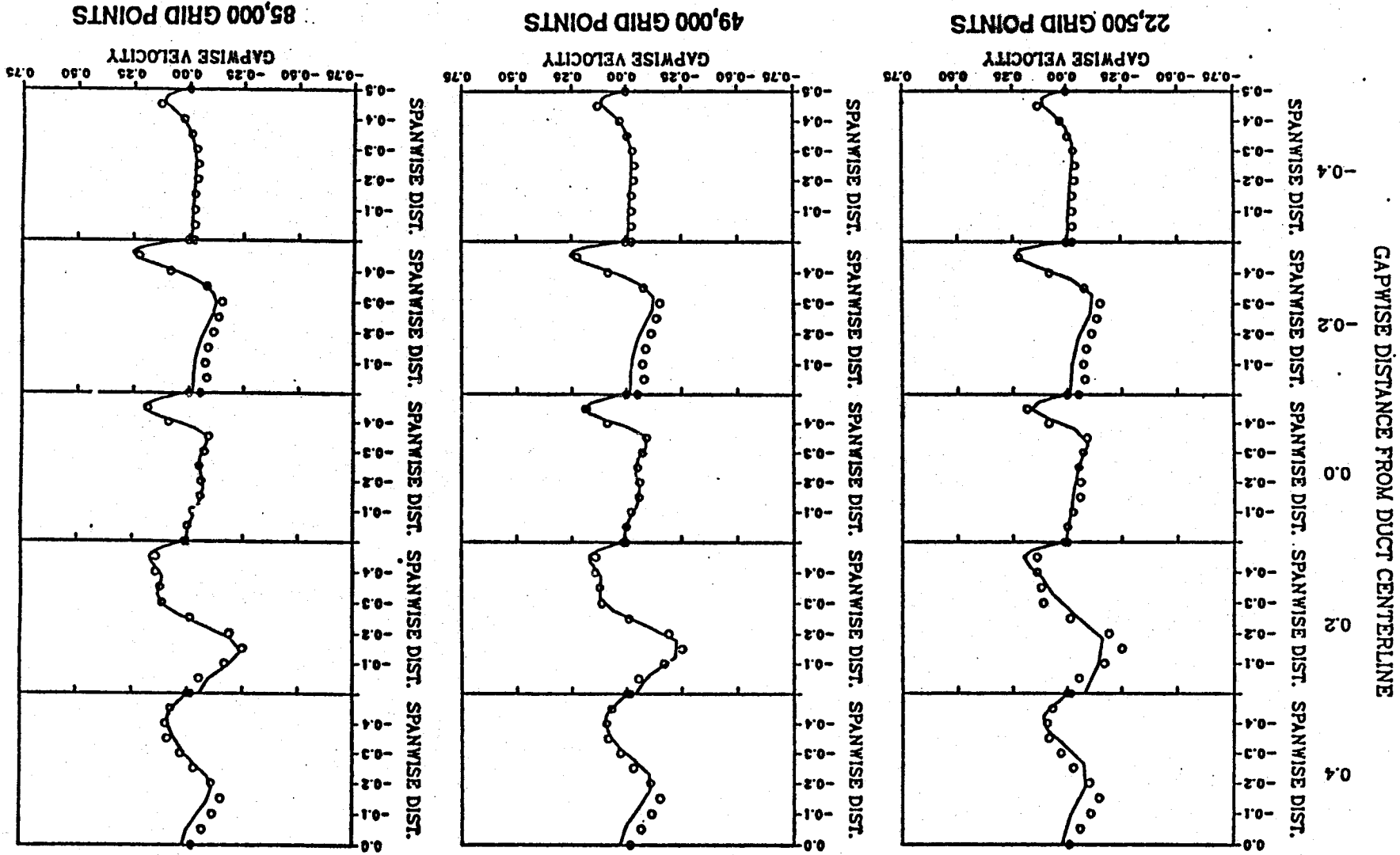
Separate Velocity Contours - INSOXU



STREAMWISE VELOCITY PROFILES - FDNS3D



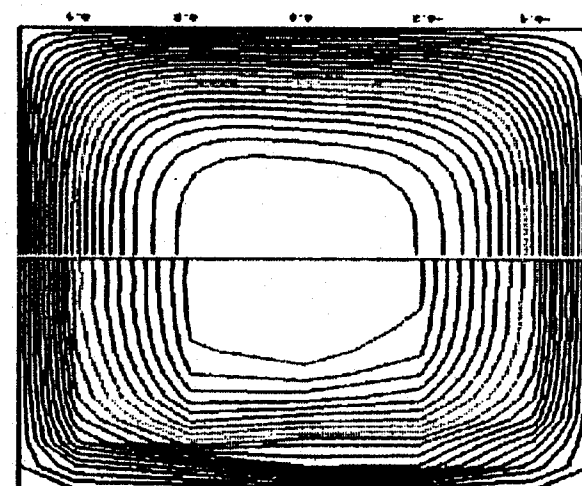
GAPWISE VELOCITY PROFILES - FDNS3D



0.25 H Past the 90° Bend Exit

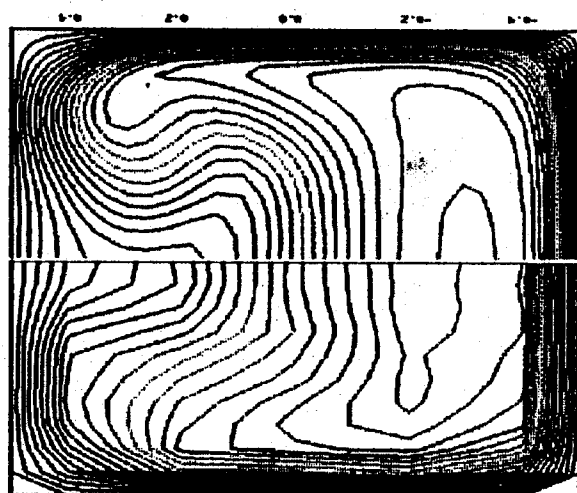
272

Streamline Velocity Contours - FOMSD



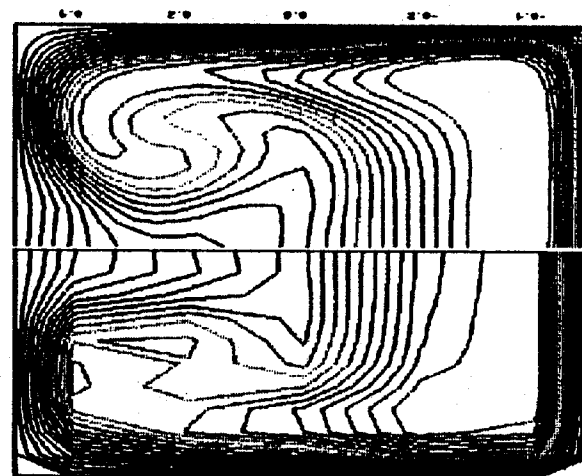
EXPANSION
COMPRESSION

$H = -0.25$



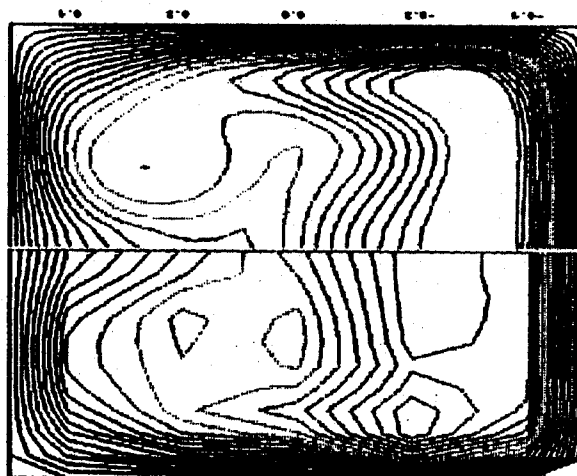
EXPANSION
COMPRESSION

$\Theta = 60 \text{ degrees}$



EXPANSION
COMPRESSION

$H = 0.25$

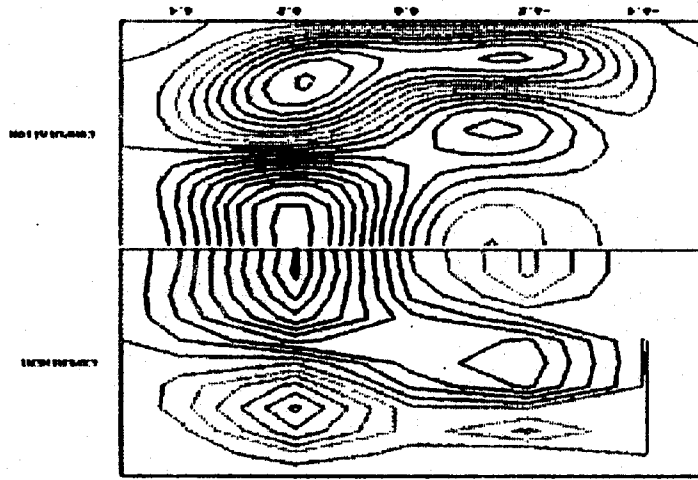


EXPANSION
COMPRESSION

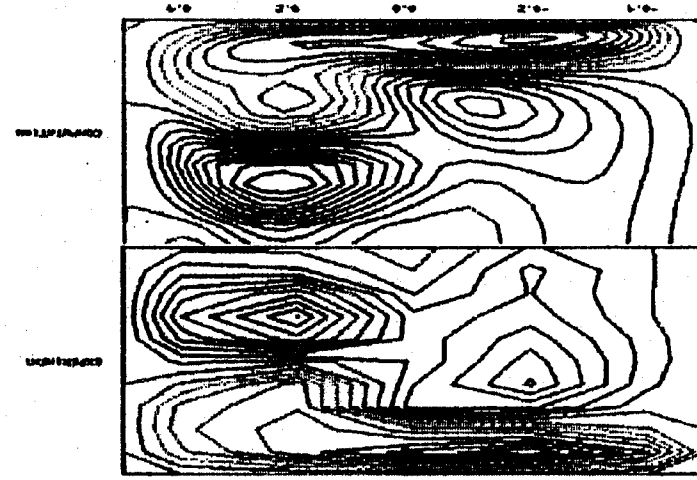
$H = 2.5$

274

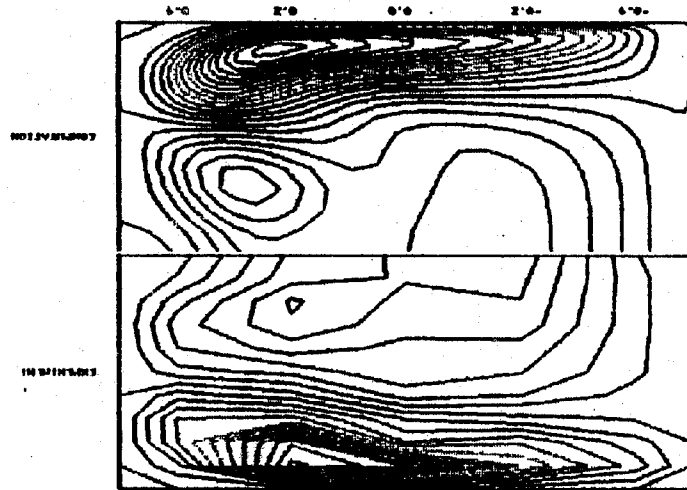
$x = 2.5 H$



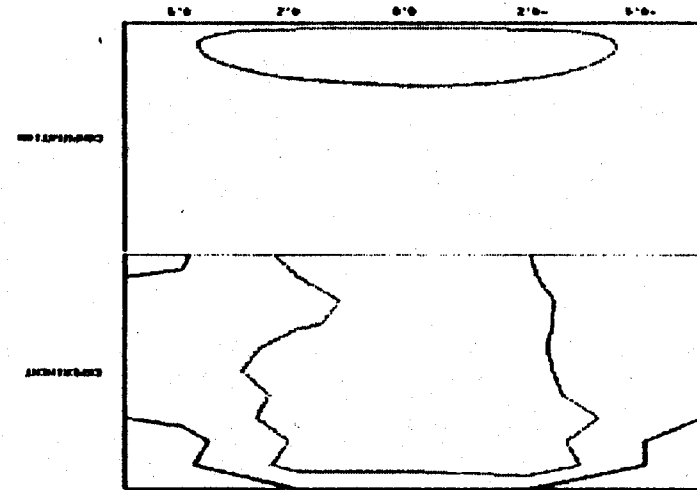
$x = .25 H$



theta = 60 degrees



$x = -.25 H$



Experimental Velocity Contours - FONS30

COMPUTER RESOURCE REQUIREMENTS

*** ALL CALCULATIONS WERE RUN ON THE CRAY XMP-416 AT MSFC**

● INS3D

- 17 WORDS/GRID POINT OF MEMORY**
- 25.0 MICRO SEC/GRID POINT/ITERATION OF CPU TIME**
- REQUIRED 3.0 MW OF MEMORY AND 1.2 CPU HOURS FOR THE CALCULATION WITH THE DENSEST GRID IN THIS STUDY BASED ON 1000 ITERATIONS**

● INS3DLU

- 74 WORDS/GRID POINT OF MEMORY**
- 11.5 MICRO SEC/GRID POINT/ITERATION OF CPU TIME**
- REQUIRED 6.1 MW OF MEMORY AND .14 CPU HOURS FOR THE CALCULATION WITH THE DENSEST GRID IN THIS STUDY BASED ON 500 ITERATIONS**

● FDNS3D

- 46 WORDS/GRID POINT OF MEMORY**
- 127.4 MICRO SEC/GRID POINT/ITERATION OF CPU TIME**
- REQUIRED 3.7 MW OF MEMORY AND 3.0 CPU HOURS FOR THE CALCULATION WITH THE DENSEST GRID IN THIS STUDY BASED ON 1000 ITERATIONS**

SUMMARY/CONCLUSIONS

- **VALIDATION OF INS3D, INS3DLU, AND FDNS3D FOR LAMINAR FLOW THROUGH A 90° DUCT BEND HAS BEEN COMPLETED**
- **AGREEMENT BETWEEN PREDICTED AND MEASURED AXIAL FLOW WAS GOOD FOR EACH CODE (PROVIDED THE GRIDS WERE DENSE ENOUGH TO ADEQUATELY RESOLVE ALL FLOW COMPONENTS).**
- **FOR ADEQUATE FLOW RESOLUTION, INS3DLU AND FDNS3D REQUIRED GRIDS OF 85,000 GRID POINTS. INS3D REQUIRED 100,000 TO 175,000 GRID POINTS.**
- **INS3DLU SHOWS A MARKED IMPROVEMENT OVER INS3D IN COMPUTATIONAL SPEED. FDNS3D A MORE
GENERAL CODE, IS THE LEAST EFFICIENT IN TERMS OF COMPUTATIONAL SPEED OF THE
THREE CODES**

TURBOMACHINERY CODE VALIDATION

<u>CODE</u>	<u>APPLICATION</u>	<u>PRINCIPAL INVESTIGATORS</u>
MTSBL/STAN5	UTRC LSRR FIRST STAGE SSME HPFTP (FPL) FIRST STAGE SSME HPFTP TTA FIRST STAGE (MSFC AND CALSPAN TESTS)	HELEN MCCONNAUGHEY HELEN MCCONNAUGHEY JOE RUF
FDNS3D	SSME HPFTP TTA FIRST STAGE (MSFC TESTS)	HELEN MCCONNAUGHEY
ROTOR1	UTRC LSRR FIRST STAGE SSME HPFTP (FPL) FIRST STAGE SSME HPFTP TTA FIRST STAGE (MSFC TESTS)	LISA GRIFFIN LISA GRIFFIN, HELEN MCCONNAUGHEY LISA GRIFFIN
ROTOR3	KOPPER'S CASCADE HODSON'S CASCADE * UTRC LSRR FIRST STATOR * LANGSTON'S CASCADE GGGT FIRST STATOR SSME HPFTP TTA FIRST STAGE (CALSPAN TESTS) UTRC LSRR FIRST ROTOR HEAT TRANSFER	LISA GRIFFIN/PRATT & WHITNEY LISA GRIFFIN/PRATT & WHITNEY LISA GRIFFIN/PRATT & WHITNEY LISA GRIFFIN/PRATT & WHITNEY LISA GRIFFIN LISA GRIFFIN LISA GRIFFIN

INTRODUCTION/OBJECTIVES

- **OVERALL OBJECTIVE - VERIFICATION OF THE UNSTEADY, 3D, VISCOUS FLOW PREDICTION CAPABILITIES OF THE ROTOR3 CODE**
- **PHASE I OBJECTIVE - VERIFICATION OF THE PERFORMANCE AND HEAT LOAD PREDICTION CAPABILITIES OF ROTOR3 THROUGH COMPARISON WITH BENCHMARK DATA**
- **PHASE II OBJECTIVE - VERIFICATION OF THE UNSTEADY FLOW PREDICTION CAPABILITIES OF ROTOR3 THROUGH COMPARISON WITH UNSTEADY FLOW DATA**
- **PHASE III OBJECTIVE - PREDICTION OF THE FLOWFIELD IN A ROCKET ENGINE TURBINE (SSME OR ADVANCED TURBINE) WITH ROTOR3 AND COMPARISON OF THE PREDICTION WITH EXPERIMENTAL DATA**

APPROACH

- **MODIFICATIONS WERE MADE TO ROTOR3 TO**
 - **ENHANCE THE CODE'S CAPABILITIES**
 - **DECREASE THE AMOUNT OF RUN TIME NECESSARY FOR PHASE I CODE VERIFICATION**
- **CODE MODIFICATIONS**
 - **CREATION OF CASCADE VERSION OF ROTOR3 WITH WHICH TO VERIFY THE CODE'S PREDICTIVE CAPABILITIES IN STEADY FLOWS FOR WHICH DETAILED DATA IS AVAILABLE**
 - **INCORPORATION OF INLET BOUNDARY LAYER SPECIFICATION**
 - **INCORPORATION OF HEAT TRANSFER PREDICTION CAPABILITY**
 - **INCORPORATION OF TRANSITION LOCATION SPECIFICATION**

4-4

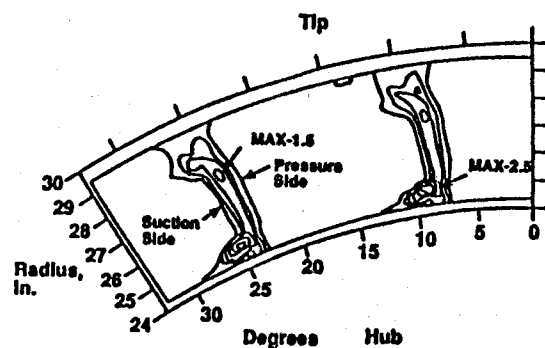
APPLICATION CONFIGURATION

- **UNITED TECHNOLOGIES RESEARCH CENTER LARGE SCALE ROTATING RIG (LSRR)
FIRST STATOR**
 - **3D ANNULAR CONFIGURATION**
 - **COMPARISON BETWEEN PREDICTIONS AND DATA USED TO VERIFY HEAT TRANSFER,
SECONDARY FLOW, AND PERFORMANCE PREDICTION CAPABILITY**
- **LANGSTON'S CASCADE**
 - **3D PLANAR CONFIGURATION**
 - **TWO DIFFERENT INLET BOUNDARY LAYER PROFILES**
 - **COMPARISONS BETWEEN PREDICTIONS AND DATA USED TO ASSESS CODE
SENSITIVITY TO INLET BOUNDARY CONDITIONS**

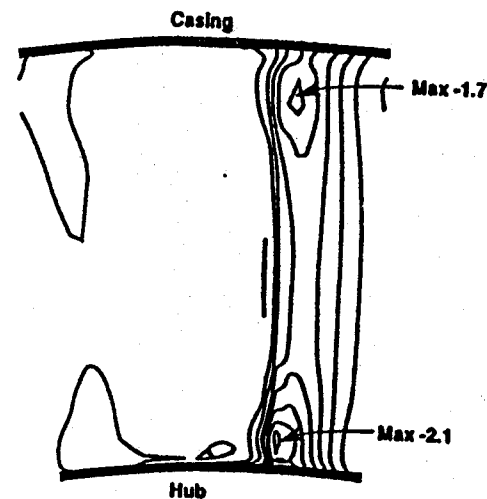
SOLUTION METHODOLOGY

- 3D UNSTEADY, THIN-LAYER, COMPRESSIBLE NAVIER-STOKES
- FACTORED, ITERATIVE, IMPLICIT ALGORITHM, THIRD-ORDER ACCURATE UPWIND DIFFERENCING SCHEME
- MODIFIED BALDWIN-LOMAX TURBULENCE MODEL, FULLY TURBULENT FLOW ASSUMED
- ADIABATIC SURFACES ASSUMED
- UNIFORM INLET CONDITIONS
- OVERLAID AND PATCHED O- AND H-TYPE GRIDS, ROTOR H-GRID SLIDES PAST STATOR H-GRID AND OUTER CASING

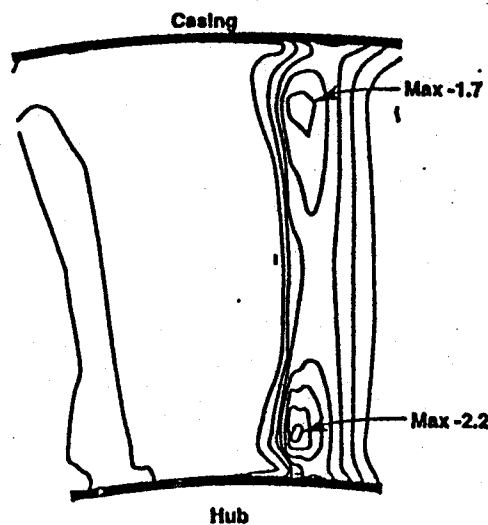
TOTAL PRESSURE LOSS CONTOURS DOWNSTREAM OF THE LSRR FIRST STATOR



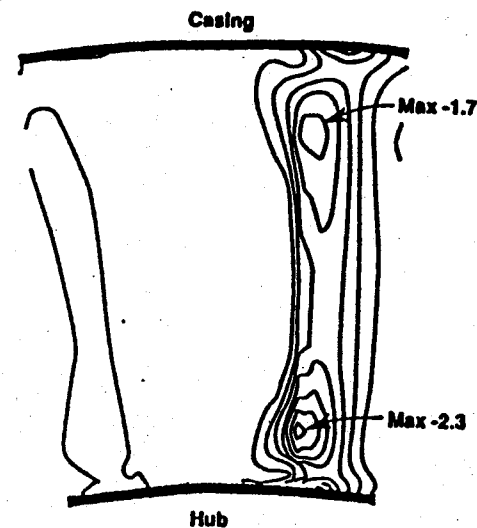
MEASURED



COARSE GRID/UNIFORM INLET

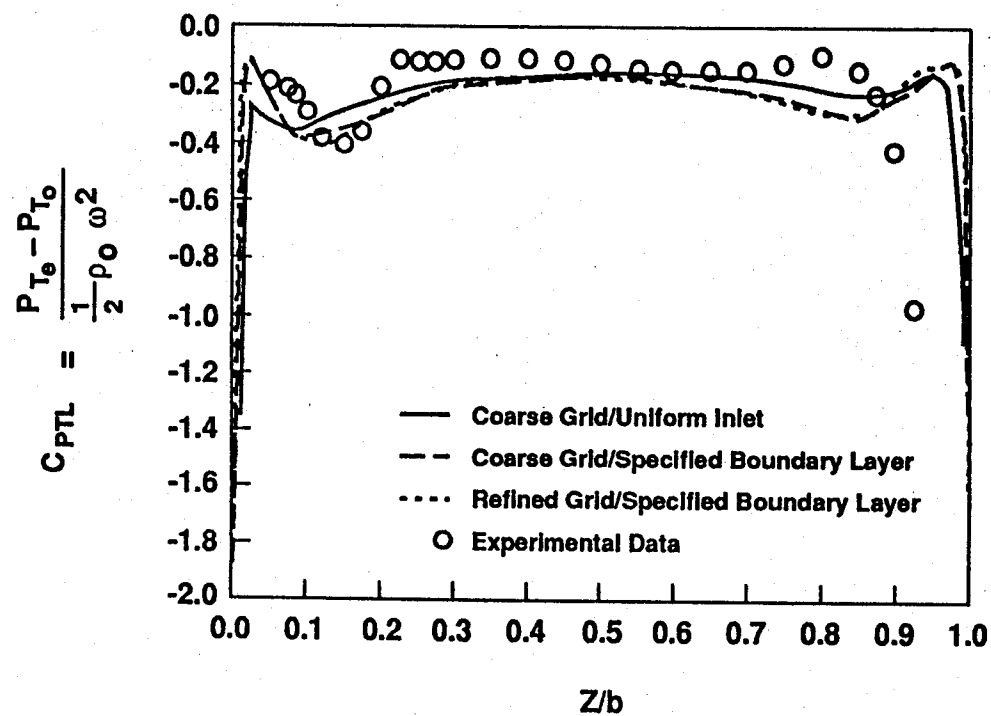


COARSE GRID/PREScribed INLET BOUNDARY LAYER

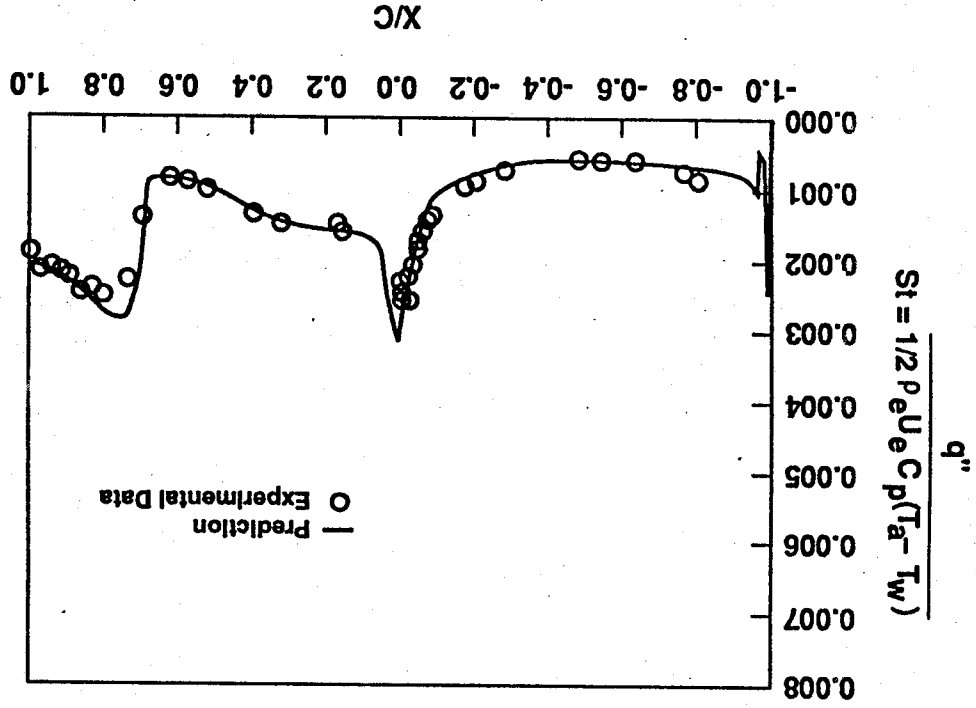


REFINED GRID/PREScribed INLET BOUNDARY LAYER

GAP AVERAGED TOTAL PRESSURE LOSS DOWNSTREAM OF THE LSSR FIRST STATOR

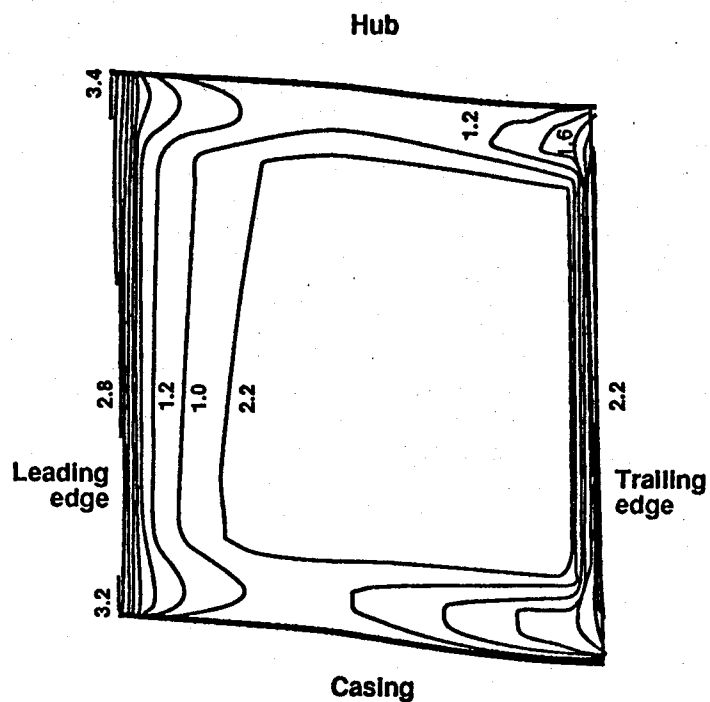


Stanton Numbers At The LSRR First Stator Midspan

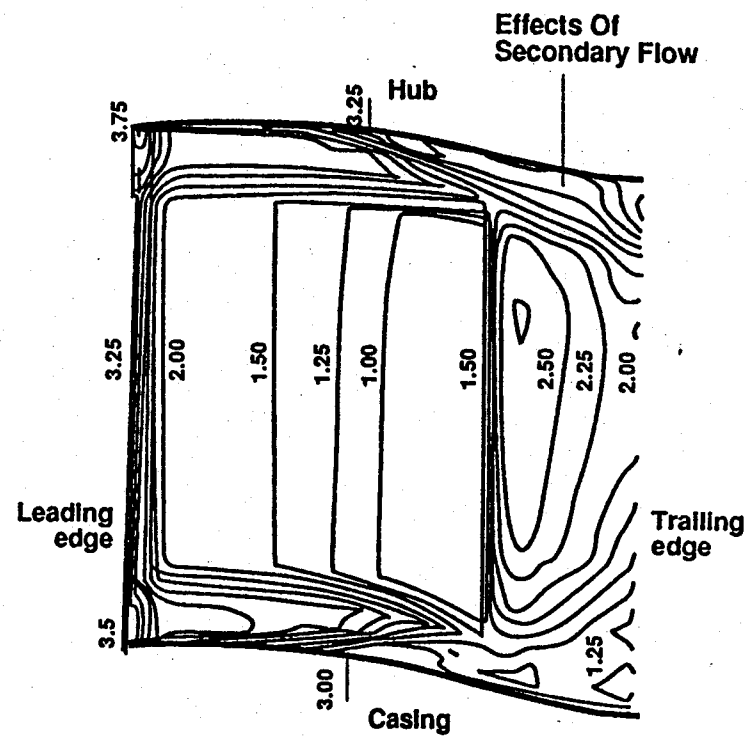


284

PREDICTED STANTON NUMBER CONTOURS FOR THE LSRR FIRST STATOR

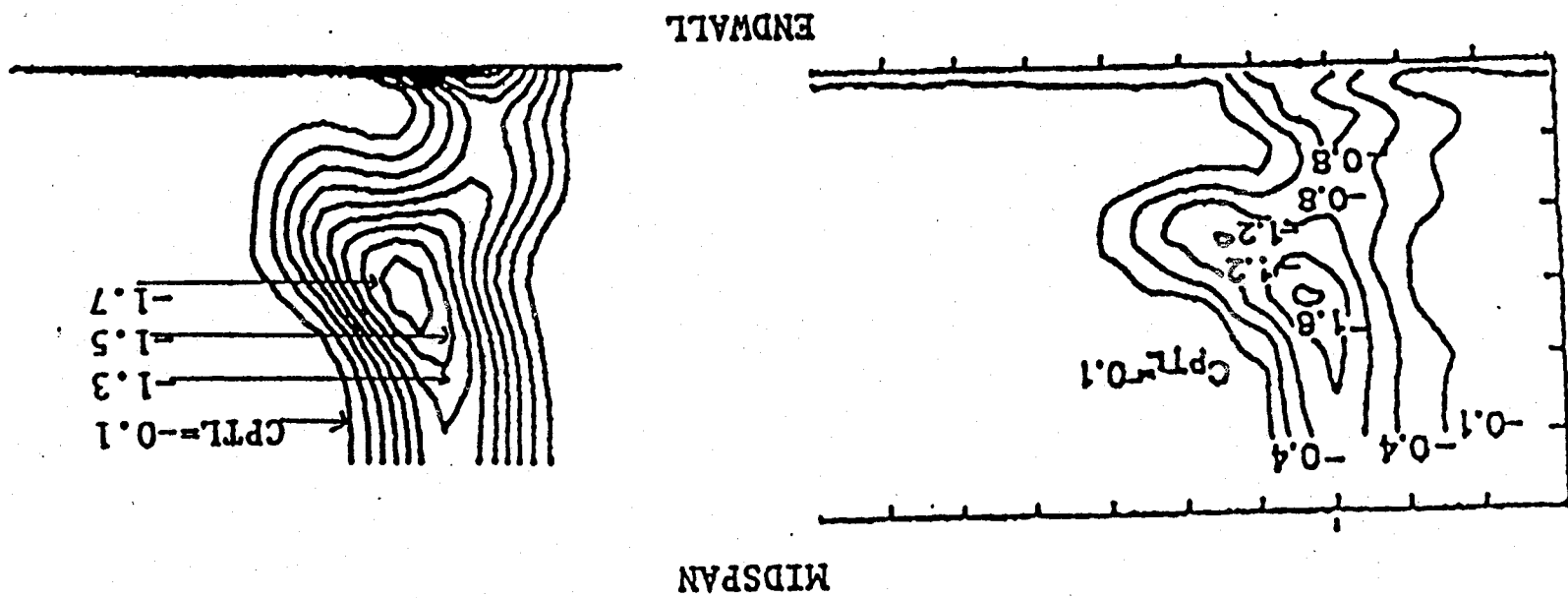


PRESSURE SURFACE

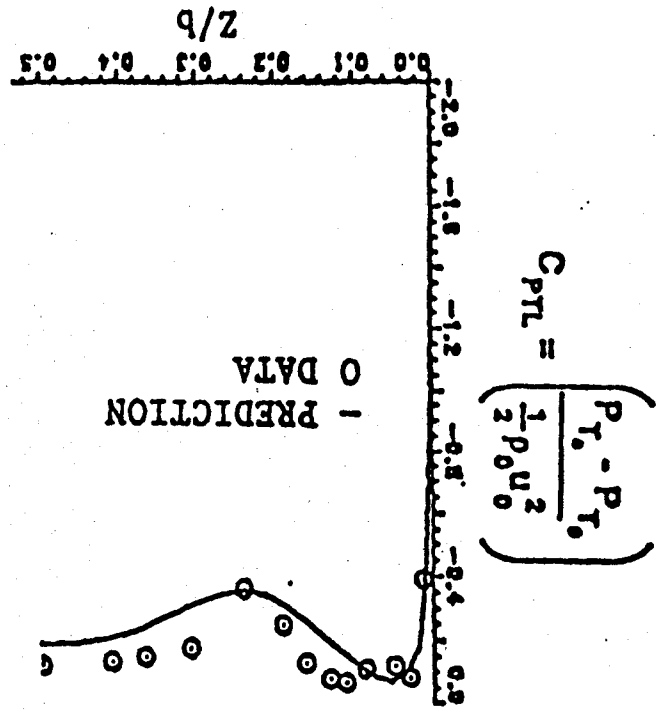


SUCTION SURFACE

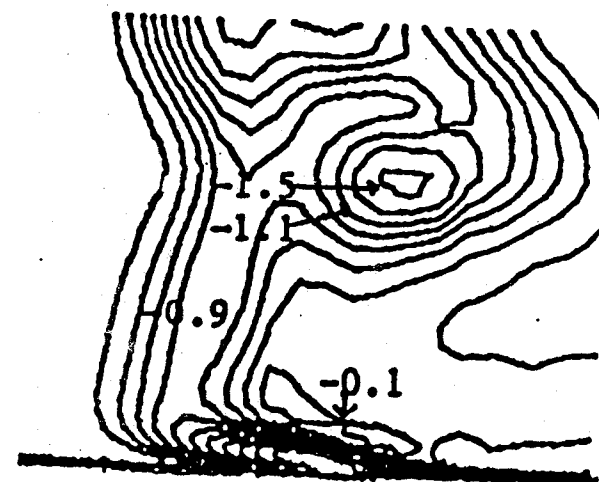
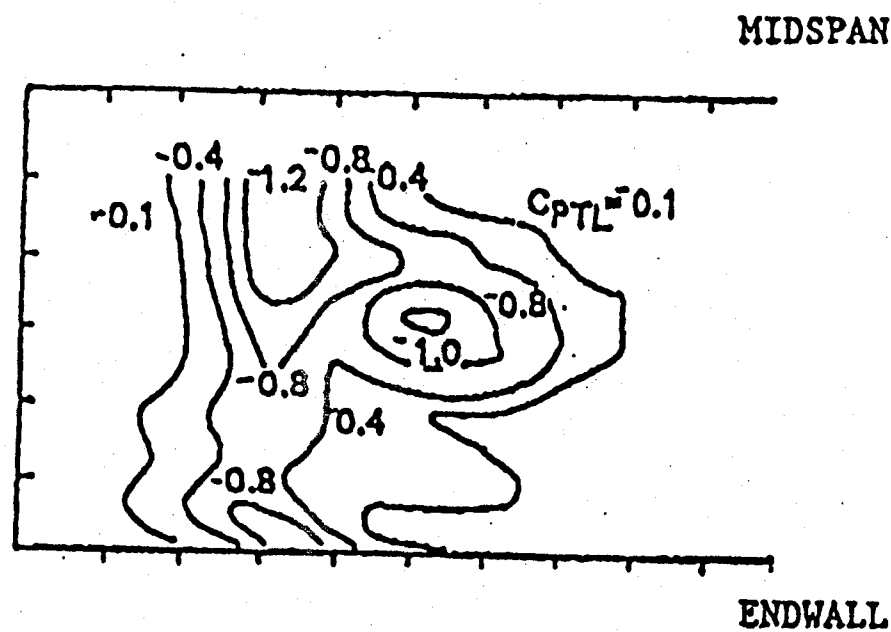
TOTAL PRESSURE LOSS CONTOURS DOWNSTREAM OF LANGSTON'S CASCADE, THIN INLET BOUNDARY LAYER



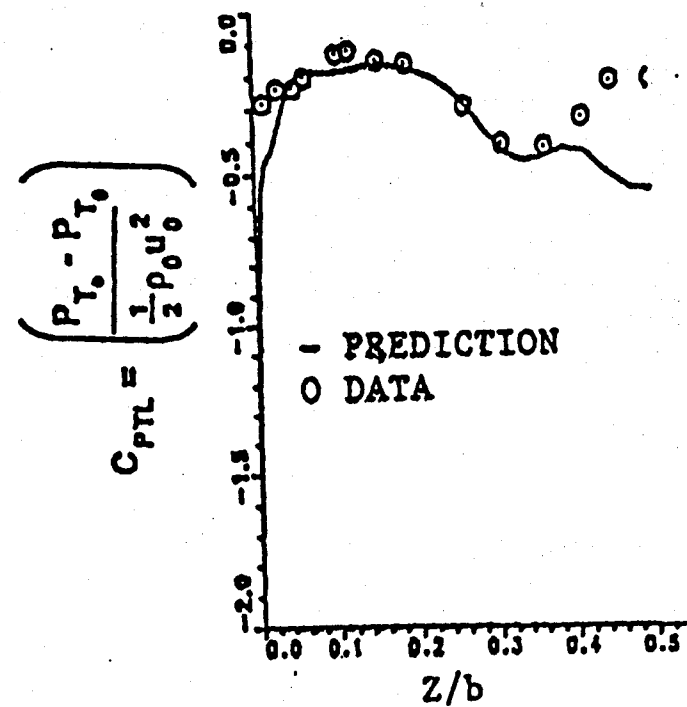
TOTAL PRESSURE LOSS THROUGH LANGSTON'S CASCADE, THIN INLET BOUNDARY LAYER



TOTAL PRESSURE LOSS CONTOURS DOWNSTREAM OF LANGSTON'S CASCADE, NOMINAL INLET BOUNDARY LAYER



TOTAL PRESSURE LOSS THROUGH LANGSTON'S CASCADE, NOMINAL INLET BOUNDARY LAYER



RESOURCE REQUIREMENTS

- CALCULATIONS RUN ON THE CRAY X-MP 416 AND CRAY X-MP-280
- CALCULATION REQUIREMENTS DEPENDENT UPON INLET CONDITION AND GRID DENSITY
- LSRR CALCULATIONS STARTED FROM FREESTREAM
- LSRR CALCULATION WITH REFINED GRID AND SPECIFIED INLET BOUNDARY LAYER PROFILE REQUIRED 20 CPU HOURS AND 4.0×10^6 WORDS OF CORE MEMORY PLUS 6.4×10^6 WORDS OF SSD
- LANGSTON'S CASCADE, THIN INLET BOUNDARY LAYER AND FINE GRID CASE, REQUIRED 4.0×10^6 WORDS OF CORE MEMORY AND 3.7×10^6 WORDS OF SSD. A CONVERGED SOLUTION REQUIRED 16 CPU HOURS WHEN STARTED FROM FREESTREAM CONDITIONS AND 7 CPU HOURS WHEN STARTED FROM AN EULER SOLUTION.
- LANGSTON CASCADE, NOMINAL INLET BOUNDARY LAYER AND FINE GRID CASE, REQUIRED 4.0×10^6 WORDS OF CORE MEMORY PLUS 5.8×10^6 WORDS OF SSD. A CONVERGED SOLUTION REQUIRED 12 CPU HOURS WHEN STARTED FROM AN EULER SOLUTION.

DISCUSSION SUMMARY

- **ACCURACY OF TOTAL PRESSURE LOSS PREDICTION TIED TO GRID DENSITY AND INLET BOUNDARY CONDITION**
 - **OVERPREDICTION OF MIDSPAN LOSS FOR EACH CASE. OVERPREDICTION INCREASES AS SECONDARY FLOW INCREASES (TURBULENT MIXING?)**
 - **UNDERPREDICTION OF CASING BOUNDARY LAYER FOR ANNULAR CASCADE (CURVATURE EFFECTS?)**
- **PREDICTED MIDSPAN HEAT TRANSFER SHOWS EXCELLENT AGREEMENT WITH DATA. COMPUTED STANTON NUMBER CONTOURS DISPLAY FEATURES CONSISTENT WITH THE FLOW**
- **HIGH RESOLUTION SMOOTH RESULTS TIED TO VERY FINE SPATIAL RESOLUTION AND, CONSEQUENTLY, LONG RUN TIMES. IMPROVED INITIALIZATION, SUCH AS STARTING THE CALCULATION WITH AN EULER SOLUTION, CAN REDUCE RUN TIME CONSIDERABLY.**

WORKING GROUP REGISTRATION

LOW SPEED

omit
to
EM?

<u>Name</u>	<u>Affiliation</u>
Gordon Pickett	Pratt & Whitney
Robert Simoneau	NASA Lewis
Scott Kjelgaard	Nasa Langley
Hugh Shin	GE Aircraft Engines
Don Dietrich	GE Aircraft Engines
Wei Tseng	Naval Air Development Center
Michael Crawford	U. of Texas, Austin
Terry Simon	U. of Minnesota
Mounir Ibrahim	Cleveland State U.
Ted Reyhner	Boeing Commercial Airplanes
Ed Schairer	NASA Headquarters
Mark Potapczuk	NASA Lewis
Lisa Griffin	NASA Marshall
Fred Gessner	U. of Washington
Reda Mankbadi	NASA Lewis
Michael Hathaway	NASA Lewis

• OUR GROUP ACCEPTED THE "BRADLEY"
DEFINITION OF VALIDATION

• VALIDATION PROGRAMS

- BACKWARD STEP - A
- TRANSITION DUCT - B
- TURN AROUND DUCT - C
- TRANSONIC CASCADE - D

IMPROVEMENTS:

- (A) Duplicate geometry
Unsteady Meas.
- (B) Needs stronger CFD input
- (C) Unsteady Meas
- (D) Duplicate geometry (##'s)

ALL

Better Initial Conditions

- ADDITIONAL BENEFIT

VALIDATION DEFINITION PROVIDED
A GOOD VISION FOR OTHER CFD PROGRAMS
(CAT. A, B, & C)

EXAMPLES:

- CENTRIFUGAL COMPRESSOR PROG.

- GENERIC GAS GENERATOR TURBINE

⇒ ULTIMATE GOAL IS VALIDATION
OF LOSES (i.e. CAT. D)

⇒ INTERMEDIATE GOAL HAS TO BE
CALIBRATION FOR COMPLEX (ENGINEERING)
TYPE FLOWS

----- AND THAT'S O.K.

- PROGRESS TO DATE ON CFD RELATED PROGRAMS IS GOOD

- INSTRUMENTATION DEVELOPMENT

- INCREASED EMPHASIS ON B.C.'s

FUTURE NEEDS - GENERAL

- IMPROVED TEAMING

- EXPERIMENTER

- CFD'er

- FLUID FLOW MODELLER

- CLEARER UNDERSTANDING AND/OR QUANTIFICATION OF VALIDATION

- IMPROVE "REWARDS" AT NASA FOR VALIDATION ACTIVITY

FUTURE NEEDS — SPECIFIC

- FLOWS WITH TRANSITION AFFECTING:

- BOY. LAYERS
- LARGE SCALE FLOW FEATURES

[MINORITY POSITION:
"UNSTEADY FLOW" SHOULD REPLACE
"TRANSITION"]

- REACTING FLOWS FOR COMBUSTORS
- ROUGHNESS EFFECTS
- REVIEW INVENTORY OF EXISTING PROGRAMS
DIRECTED TOWARDS CFD VALIDATION
 - SEPARATION BUBBLE EXP. LUTR,
 - COMPRESSOR CASCADE (PENN STATE,
 - HOST PROGRAMS
 -
 -

- HIGH "RATES-OF-STRAIN" FLOW FIELDS

- - - - -

*! CALIBRATION - - - - - VALIDATION !! *

L.S.-4

WORKING GROUP REGISTRATION

HIGH SPEED

<u>Name</u>	<u>Affiliation</u>
Joe Marvin	NASA Ames
Jerry Wood	NASA Lewis
Tony Ingraldi	NASA Langley
Kevin Early	GE Aircraft Engines
Walt Sturek	Army Ballistic Research Laboratory
Bill Compton	NASA Langley
Alan Epstein	MIT
John Malone	NASA Langley
August Verhoff	McDonnell Aircraft Company
Essam Atta	GE Aircraft Engines
Frank Spaid	McDonnell Douglas Research Laboratory
Warren Hingst	NASA Lewis

General Observations/Comments

- CFD/EFB Cooperation
essential
needs continued emphasis
- Strong need for standard data sets - NASA
 - Flow Phenomena - critical now!
 - Components
 - format for archiving
 - Hard Copy - classic test cases
 - floppy/electronic - near term
- NASA responded positively to
Bradley Committee recommendations
 - small step, however
- reporting of Validation effort is
recommended on periodic basis

To
Community

CFD Validation - Hi Speed

Focus : Hi Speed Civil Transport.
transonic \rightarrow supersonic

CFD Requirements - Propulsion
Inlet

Nozzles

Turbomachinery

Afterbody

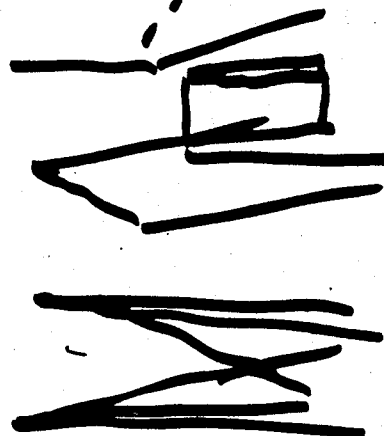
Inlets

Key Physics Issues

current program

Shock - Boundary layer Interactions

3-d



compression

glancing

crossing

Needed -

Boundary layer control exp's.

Bleed, blowing . . .

unsteadiness

Nozzles

Transition Sections

Current
 round to rectangular
 cross flow initial conditions

needed
 upstream conditions representing
 vorticity

unsteady flow (15%)

Thermal gradients

measurement of downstream boundary
 conditions

Nozzles

current - not much

needed - jet mixing at representative
 scale . . . Noise, shock noise

Rotating Machinery UNRESOLVED PROBLEMS

1. Transition & Reattachment
2. Shock - B.L. Interaction

$$M = 1 \sim 3$$

- 3. Rotating B.L.'s
- 4. Tip Leakage Vortices
- 5. Interaction of Vortices & Airfoils
- 6. Film Cooling
- 7. Unsteady 3-D BLADE Row Interactions
- 8. Heat transfer

General Comment:

1. Most often Inlet B.L. not well characterized in exp.
 2. Outlet B.C. difficult to specify
 3. Geometry must be well specified
- addressed in current program

WORKING GROUP REGISTRATION

HYPERSONICS

<u>Name</u>	<u>Affiliation</u>
Louis A. Povinelli	NASA Lewis
Irwin E. Vas	Boeing Defense and Space Group
Michael S. Holden	Calspan Corp.
William K. Lockman	NASA Ames
I. L. Bhateley	General Dynamics, Fort Worth
J. Phillip Drummond	NASA Langley
Mani Subramanian	GE Aircraft Engines
Charles R. McClinton	NASA Langley
Eli Reshotko	Case-Western Reserve U.

Hypotheses Procedure

- look at CFD validation experiments currently underway related to hypersonic aerodynamic vehicle.
- Identify new or additional requirements or needs.
- Provide recommendations for CFD validation activity

Discussion broken into following

areas:

- Value somebody
- generic building block (aero) exper.
- Fuel
- Combustors
- Nozzles
- Population System Integration

Additional considerations:

- Non air-breathing propulsion vehicles.

3.

Vehicle Databody

- All-body configuration at ARC (eligible data wing) has produced primary code calibration data.
- Mc Donnell-Douglas Gemini Option 2 (Blend - as wing body) mounted significant cross - flows on inside near inlet plane, not captured by codes.

Vehicle Forebody Tests

- Experiments in number of facilities on generic set of elliptic cones with ellipsoidal bluntness to provide validation data including:
 - 3D flow field (and boundary layers)
 - Transition (3D) (Multiple mechanisms)
 - Real gas effects
 - Tunnel flow quality & availability
 - Fluctuating non-intrusive measurements.

Building Block (Aero) Experiments

(Shock-wave BLI)

- 2D Compressor ramp shock interaction (3D)
- glancing & crossing shock interaction
- 2D Oblique/Wall layer interaction
 - film cooling
 - Transpiration cooling
- flow thrust/admiss & separation

Building Block Experiments

- Appropriate level of effort
- Sufficient detail measurements
 - related to turbulence modeling (flow - velocity flow field measurements).
- Reynolds stress components
- Density fluctuations
- Pressure fluctuations
- Velocity fluctuations

Inlets

- Revised current data base
- Consider "generic" inlet for valve - atom experiment. (M10 underway)
- Current data base in some heating adequate. Requires imp - and modeling for code - in particular for transitional shear layers.
- Experimental study of inlet unstart. In audience in modeling. 310

Inputs

- Mach 5 produced limited validation data; primarily calibration data
- Mach 1225 (MC D - gen. Opt) yielded heat transfer/pressure data. gen-ally not validation
- Mach 18 Subsonic inlet produced some flow field data (limited)

Combustors

- Experiments identified to study:
 - wall jets (film cooling)
 - co-axial jets (w & w/o reaction)
 - swept ramp injectors
 - vectored jet injection
 - high Mach, hi-enthalpy mixing & combustion
 - Effect of divergence on turbulence.

Combustion

10.

- Only limited experiments feasible.
- limited diagnostic capabilities for reacting (i.e. ~~combusting~~) flows

• Need measurements of:

Species (mean & fluctuating)
Temp
Velocity

- Structure of mixing process.
- Dense combustor shape needs

to be defined

• ~~grumman~~
kAC
AAC
8x6 kAC

• Run current code against
existing maple data

1103384

11.

Integrated System (Experiments)
 is a design issue
 related to performance
 parameters (primarily)

Not a NASA task, but
 rather industrial / NASA

Proper codes (valuated)

should work properly.
 2.5

Thm - Air breathing Hydroponics

- Considers continuum; non-cont.
- High drag, $L/D < 2$.
- Concerns with thermal protection, radiation, b.l. transition, heat gas effects
- Suggest separate group to discuss radiation issues

